

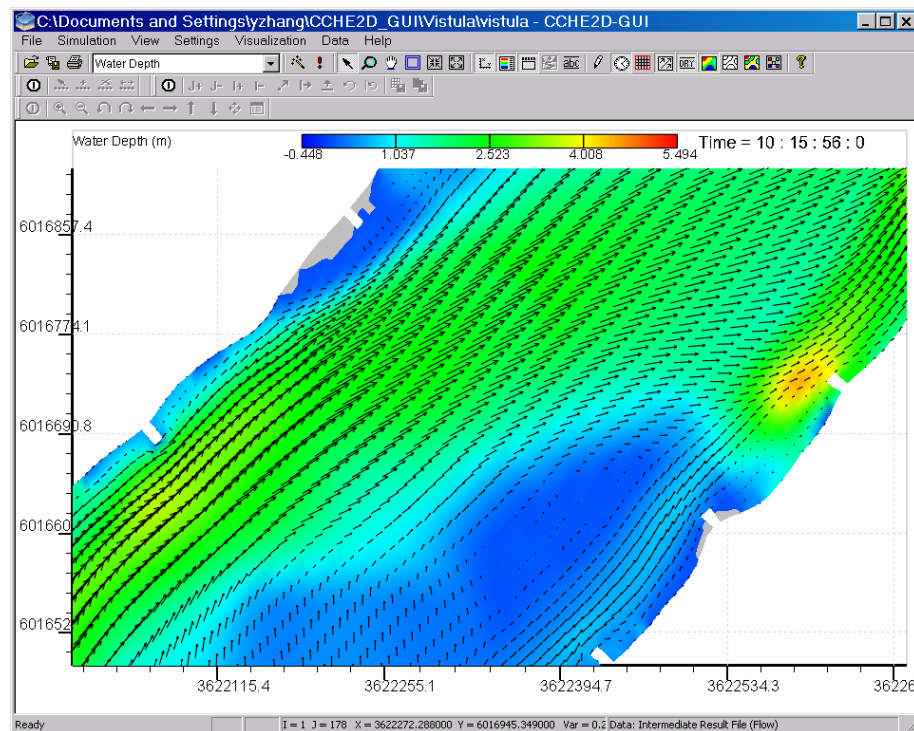
NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

CCHE2D-GUI Version 2.2 – Quick Start Guide

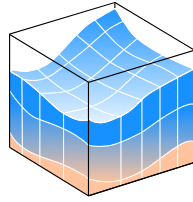
Technical Report No. NCCHE-TR-2005-04

June 2005

Yaoxin Zhang



School of Engineering
The University of Mississippi
University, MS 38677



NATIONAL CENTER FOR COMPUTATIONAL
HYDROSCIENCE AND ENGINEERING

Technical Report No. NCCHE-TR-2005-04

CCHE2D-GUI Version 2.2– Quick Start Guide

Yaoxin Zhang

Post-doctoral Research Associate

The University of Mississippi

June 2005

Table of Content

1 INTRODUCTION.....	1
1.1 PURPOSE	1
1.2 APPLICABILITY	1
1.3 USING THIS GUIDE.....	1
1.4 RELATED DOCUMENTS.....	2
2 EXAMPLE 1: STEADY FLOW SIMULATION.....	3
2.1 INTRODUCTION	3
2.2 OPEN GEO FILE	3
2.3 SET FLOW INITIAL CONDITIONS	5
2.4 SET FLOW PARAMETERS	10
2.5 SET INLET AND OUTLET BOUNDARY CONDITIONS	13
2.6 RUN CCHE2D MODEL	17
2.7 VISUALIZE FLOW RESULTS	18
3 EXAMPLE 2: STEADY FLOW SIMULATION WITH SEDIMENT TRANSPORT	25
3.1 INTRODUCTION	25
3.2 OPEN GEO FILE	25
3.3 SET FLOW PARAMETERS	27
3.4 SET SEDIMENT PARAMETERS	27
3.5 SET SEDIMENT BOUNDARY CONDITIONS	30
3.6 SET INLET AND OUTLET BOUNDARY CONDITIONS	36
3.7 SET BED MATERIAL SAMPLES	38
3.8 SET BED MATERIAL PROPERTIES	40
3.9 RUN CCHE2D MODEL	45
3.10 VISUALIZE SEDIMENT RESULTS	46
4 EXAMPLE 3: UNSTEADY FLOW SIMULATION USING SIMULATION WIZARD.	50
4.1 INTRODUCTION	50
4.2 OPEN GEO FILE	50
4.3 USE SIMULATION WIZARD	52

4.4 VISUALIZE RESULTS	61
-----------------------------	----

1 Introduction

1.1 Purpose

This Quick Start Guide is intended to provide step-by-step instructions through example applications to help new users get familiar with the CCHE2D-GUI. It keeps emphasizes on how to use CCHE2D-GUI to run numerical simulations in the interactive graphical environment. Two main examples are described in this guide to demonstrate the application of CCHE2D-GUI to practical engineering problems.

This guide will neither show all the capabilities nor describe in detail the functionality of each feature of CCHE2D-GUI. For a complete description of CCHE2D-GUI, please refer to CCHE2D-GUI Users' Manual.

1.2 Applicability

This Guide applies to CCHE2D-GUI version Beta 2.2. The current version 2.2 is NOT completely backward compatible with the previous versions. Users of previous versions should backup their data before using this version to simulate the old cases you've already done to avoid the loss or damage of data.

1.3 Using This Guide

This guide assumes that CCHE2D-GUI has already been installed and it is working properly on your computers. If you have any problems with the installation, please contact the developer for help.

This guide uses data sets of real rivers to demonstrate how to use CCHE2D-GUI for flow simulation and sediment transport simulations. The first example is the steady flow

simulation for Vistula River in Poland. The second example, the steady flow simulation with sediment transport, is continuous from the first example. The third example uses a reach of the Mississippi River in United States for unsteady flow simulation without sediment transport. By following these three examples, you should be able to set up CCHE2D for the simulations of similar cases. For more complicated cases, you need to explore the CCHE2D-GUI Users' Manual in detail. In order to simulate real cases more reasonably and correctly, please refer to the Technical Report of CCHE2D Numerical Model.

1.4 Related Documents

The documentation of CCHE2D consists of several publications designed to fulfill the needs of different audiences. They are simply listed as follows:

- “CCHE2D-GUI Quick Start Guide” is intended for the first-time users.
- “CCHE2D-GUI – Graphical Users Interface for CCHE2D Model - User's Manual” describes in detail the capabilities and How-Tos of CCHE2D-GUI.
- “CCHE2D: Two-dimensional Hydrodynamic and Sediment Transport Model for Unsteady Open Channel Flows Over Loose Bed” describes in detail the basic mathematics, numeric, hydraulics and sediment transport approaches.
- “CCHE2D Sediment Transport Model” describes in detail the governing equations, boundary conditions, numerical methods and empirical formulas of the CCHE2D non-equilibrium transport model of non-uniform sediment.
- “CCHE2D Mesh Generator Users' Manual” is aimed at how to generate computational meshes for the CCHE2D model. It is associated with a separate software: CCHE2D Mesh Generator.


2 Example 1: Steady Flow Simulation

2.1 Introduction

This chapter will illustrate how to use CCHE2D-GUI to simulate steady flow using the data from the Vistula River in Poland. All data is included as example files in the installation package of CCHE2D-GUI.

2.2 Open Geo File

The first step in getting a case ready for simulation is to generate a mesh for the domain of interest. A mesh file, a file with a “**.geo**” extension, is required by both the CCHE2D-GUI and the CCHE2D numerical model. The mesh file contains the geometry and the initial conditions of the grid, such as the (x,y) coordinates, the initial bed elevation, the initial water surface level, the bed roughness and the nodal type. For use in this guide, a mesh file is provided with the model.

- Start CCHE2D-GUI by either double-clicking the CCHE2D-GUI icon on your Windows desktop or by clicking the icon from CCHE2D-GUI group of your Windows programs in the start menu.
- Select **Open Geometry...** in the **File** menu or click  in the main toolbar. Navigate to the directory named **Vistula** and select the **Vistula.geo** file, then click **Open**.

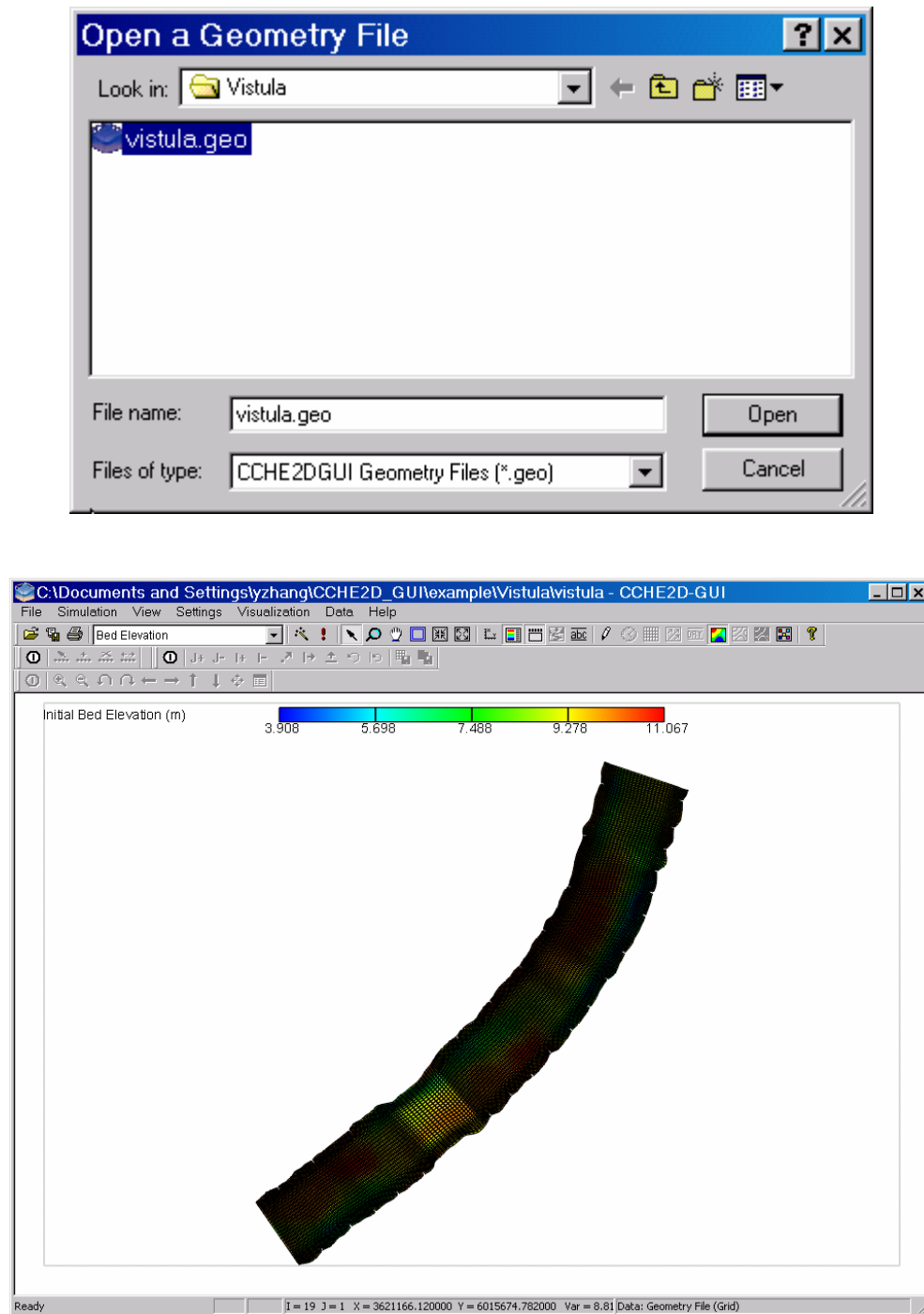


Figure 2-1

If you are not satisfied with the mesh, you can either regenerate a new mesh using CCHE2D Mesh Generator or edit the mesh for small modifications using the **Grid Editing** toolbar of CCHE2D-GUI. For details of grid editing, please refer to Chapter 4 in CCHE2D-GUI Users' Manual.

2.3 Set Flow Initial Conditions

In addition to the geometry information, the mesh file also contains the information of the initial flow conditions that include the initial water surface level, bed roughness, and initial bed elevation. You can use the CCHE2D-GUI to modify them.

- Select **Set Flow Initial Conditions...** in **Simulation** menu to activate the **Nodal Properties** dialog window.

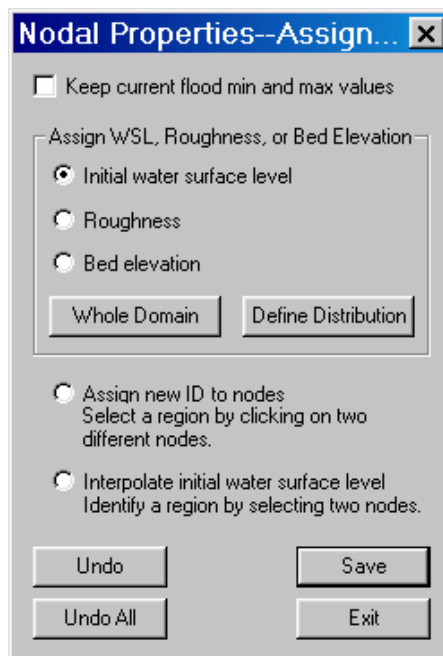


Figure 2-2

- You can select the flow variable by checking the radio button in **Assign WSL, Roughness** or **Bed Elevation**. To specify a constant value of selected variable through the entire domain, you should click **Whole Domain**; to specify a spatial

distribution, you should click **Define Distribution** and then selecting two points to define a rectangular area within the mesh.

- First select option **Initial water surface level**.
 - For this case, we choose to specify a **linear distribution** in **J direction** for initial water surface level. To identify the I and J directions of the mesh, you can move the mouse on the mesh, the nodal information including the I and J index will be displayed in the status bar. Click **Define Distribution** and then select two points along the last mesh J line. In **Assign Value** dialog, enter value 10 and click **OK**.

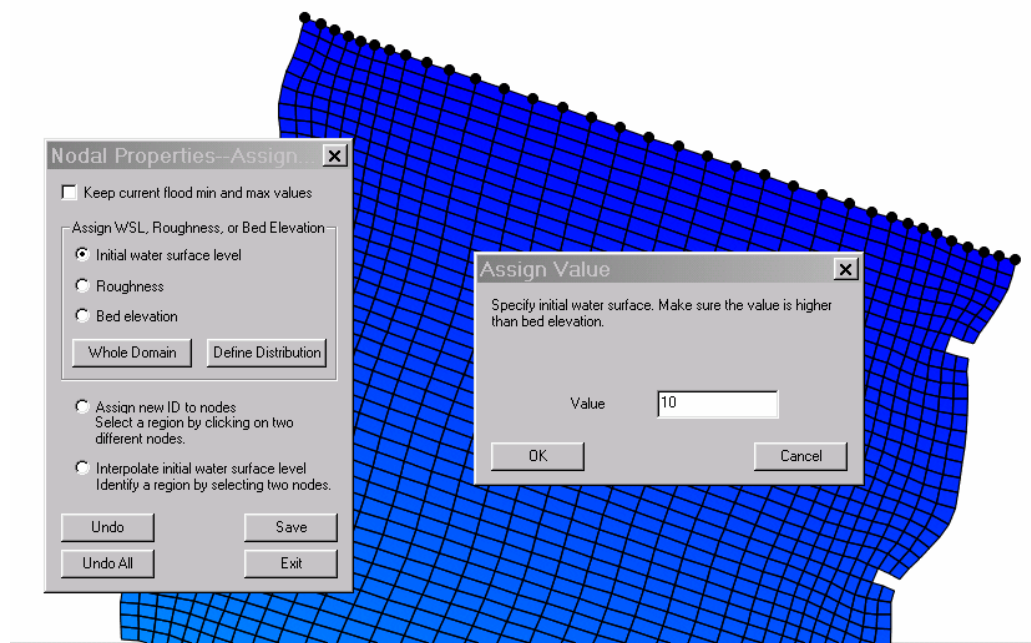


Figure 2-3

- Click **Define Distribution** again and then select two points along the first mesh J line. In **Assign Value** dialog, enter value 11.6 and click **OK**.

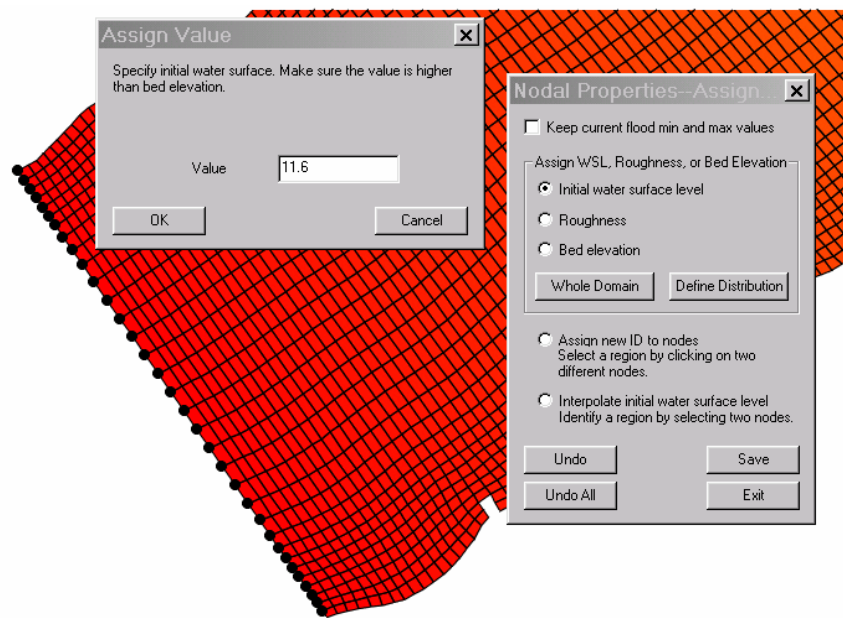


Figure 2-4

- Select the option **Interpolate initial water surface level** and then click two different points to cover the whole domain. In dialog **Initial Water Level Interpolation**, choose **Interpolate along constant I-lines**, and then click **OK**. The initial water surface level will be linearly interpolated along constant I lines.

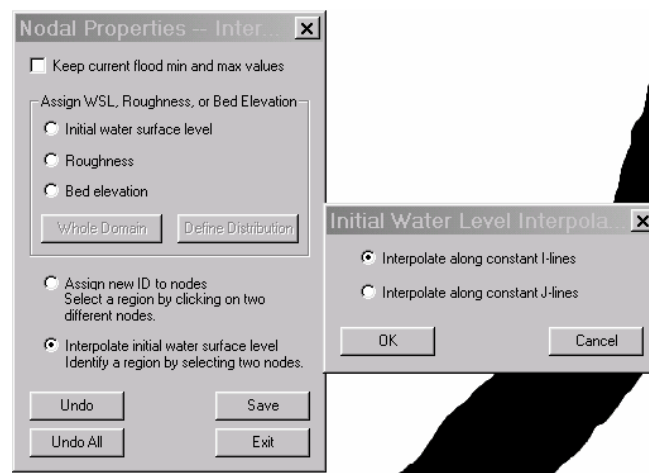


Figure 2-5

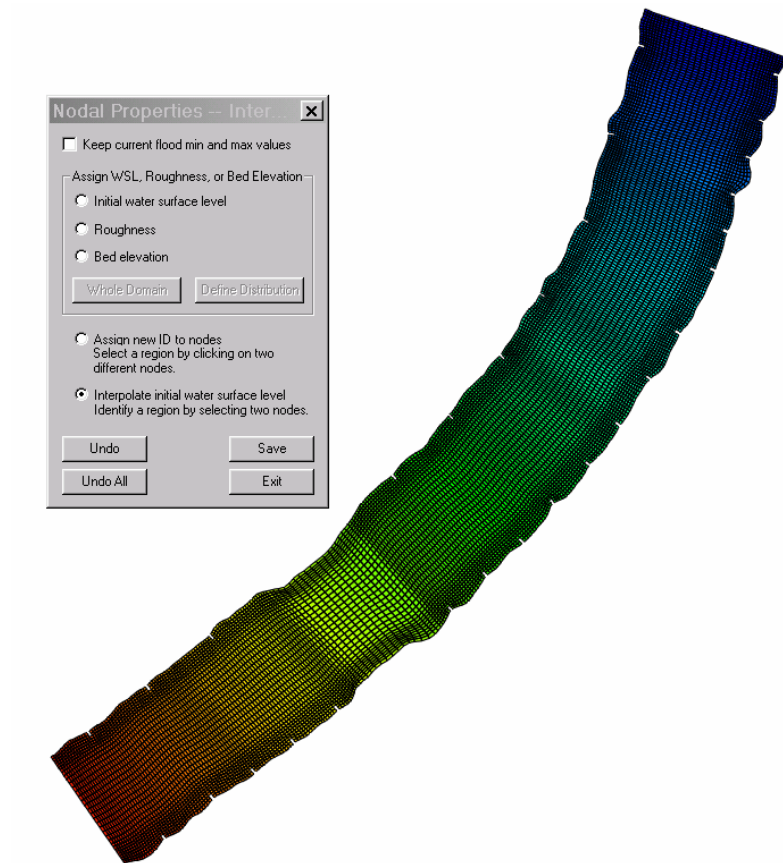


Figure 2-6

- Select option **Roughness**.
 - First click **Whole Domain** and enter 0.035 in the **Assign Value** dialog, and then click **OK**.

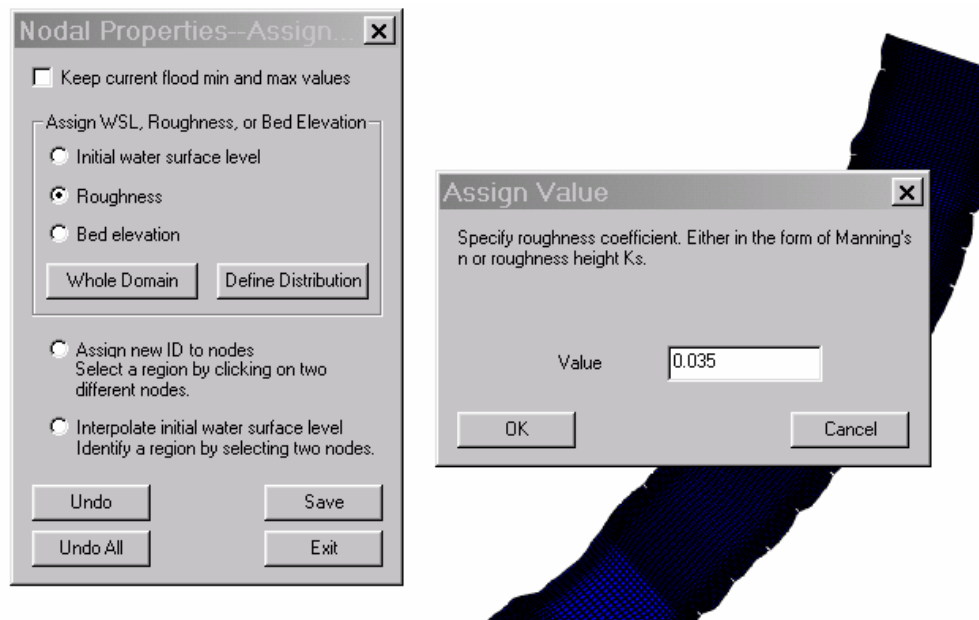
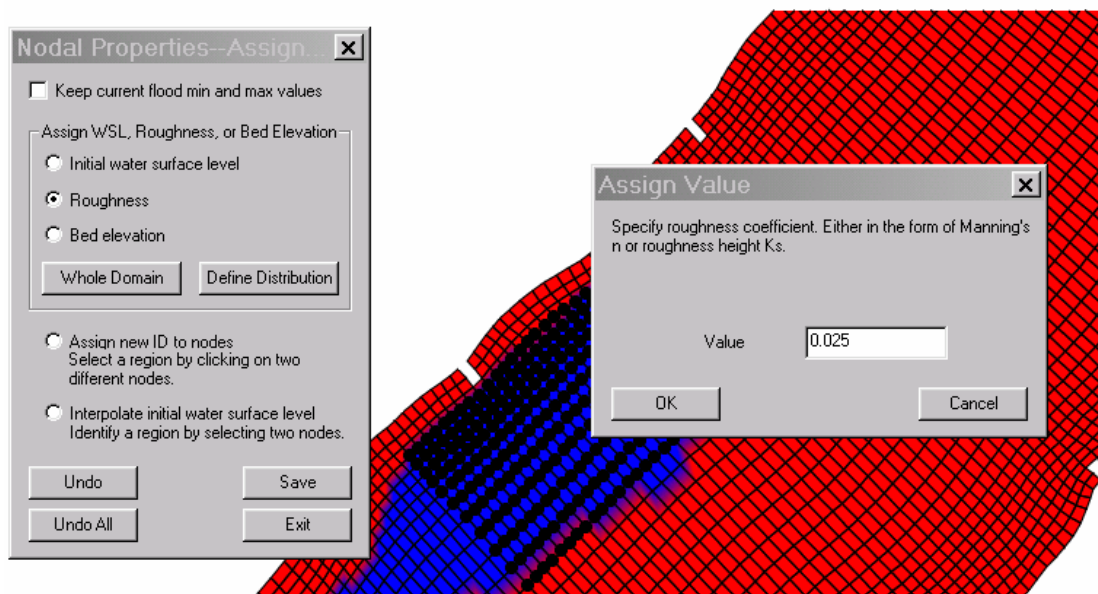


Figure 2-7

- Then click **Define Distribution** to specify another value 0.025 for several regions.



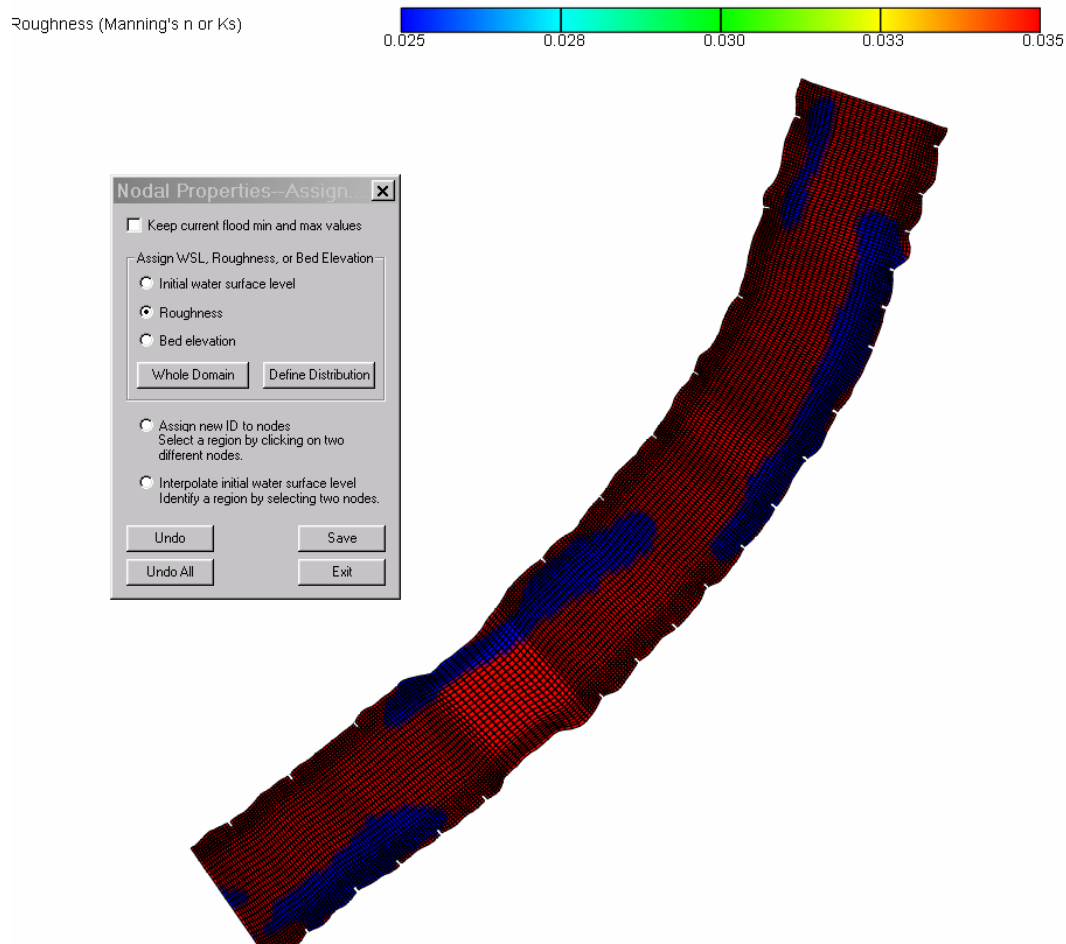


Figure 2-8

- Click **Save** to save your changes into the **geo** file. Note that the data in the geo file will be replaced by the data you just provided.

2.4 Set Flow Parameters

The flow parameters consist of **Simulation** parameters, **Bed Roughness** parameters and **Advanced** parameters.

- Select **Set Flow Parameters...** in **Simulation** menu to invoke the **Set Flow Parameters** dialog window.

- In **Simulation Parameters** page, specify the parameters as shown in Figure 2-9.

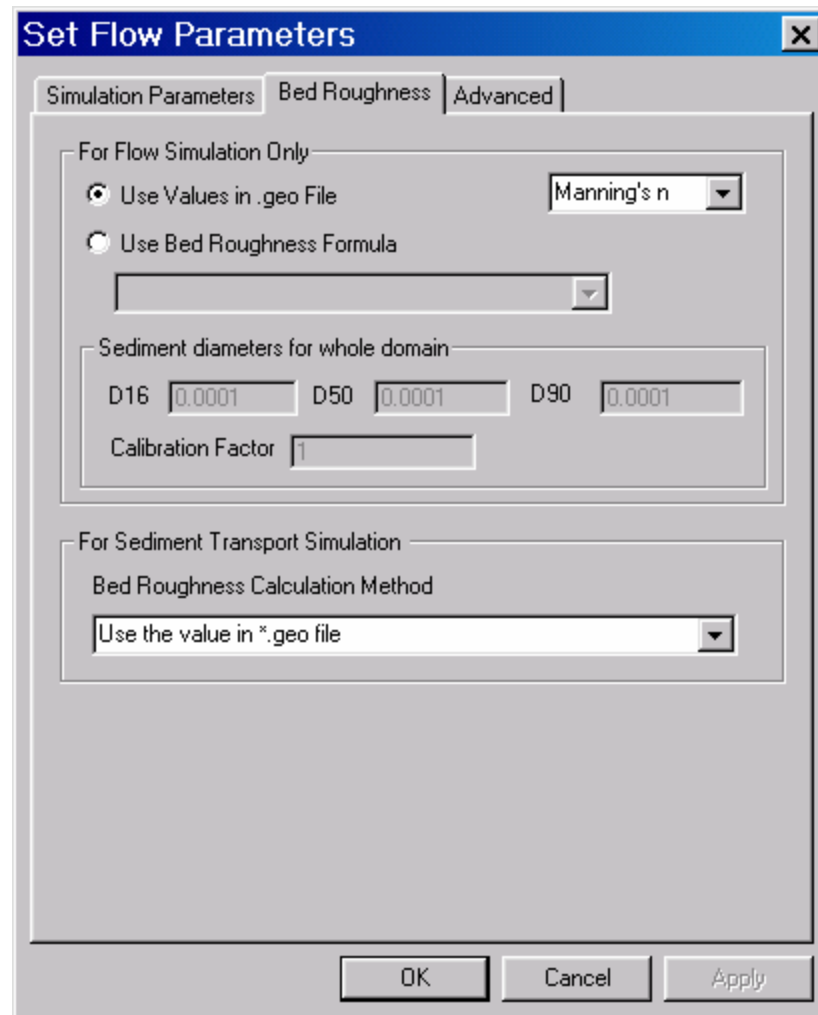
The screenshot shows the 'Set Flow Parameters' dialog box with the 'Simulation Parameters' tab selected. The dialog is divided into several sections:

- Time Step:** Simulation time (s) is 518400, Time step (s) is 10, and Total Time Steps is 51840.
- Time Steps for Output:** Intermediate File is 1000, History File is 1000, Monitor Points is 0, and Convergence is 1.
- Turbulence:** Turbulence Model Option is set to 'Parabolic Eddy Viscosity Model' (dropdown), and Turbulent viscosity coefficient is 1.
- Unsteady Flow Computation:** The checkbox 'Compute as quasi steady flow' is unchecked, and Time Steps to reach steady state is 1.
- Numerical:** Wall slipness coefficient is 0.5, Depth to consider dry (m) is 0.04, and Time Iteration Method is 'Method 1' (dropdown).

At the bottom of the dialog are three buttons: OK, Cancel, and Apply.

Figure 2-9

- In Bed Roughness page, make sure the option **Use Values in .geo File** and **Manning' n** in the group **For Flow Simulation Only** are selected. The **Bed Roughness Calculation Method** in the group **For Sediment Transport Simulation** automatically takes the same option as the one selected for flow.



The image shows a software dialog box titled "Set Flow Parameters" with a close button (X) in the top right corner. It has three tabs: "Simulation Parameters", "Bed Roughness", and "Advanced". The "Advanced" tab is currently selected. Inside the dialog, there are two main sections. The first section, "For Flow Simulation Only", contains two radio buttons: "Use Values in .geo File" (which is selected) and "Use Bed Roughness Formula". To the right of the first radio button is a dropdown menu showing "Manning's n". Below the radio buttons is another dropdown menu. The second section, "For Sediment Transport Simulation", contains a label "Bed Roughness Calculation Method" and a dropdown menu showing "Use the value in *.geo file". Between these two sections is a group box titled "Sediment diameters for whole domain" containing three input fields: "D16" with value "0.0001", "D50" with value "0.0001", and "D90" with value "0.0001". Below these is a "Calibration Factor" input field with the value "1". At the bottom of the dialog are three buttons: "OK", "Cancel", and "Apply".

Figure 2-10

- In **Advanced** page, we use the default values.

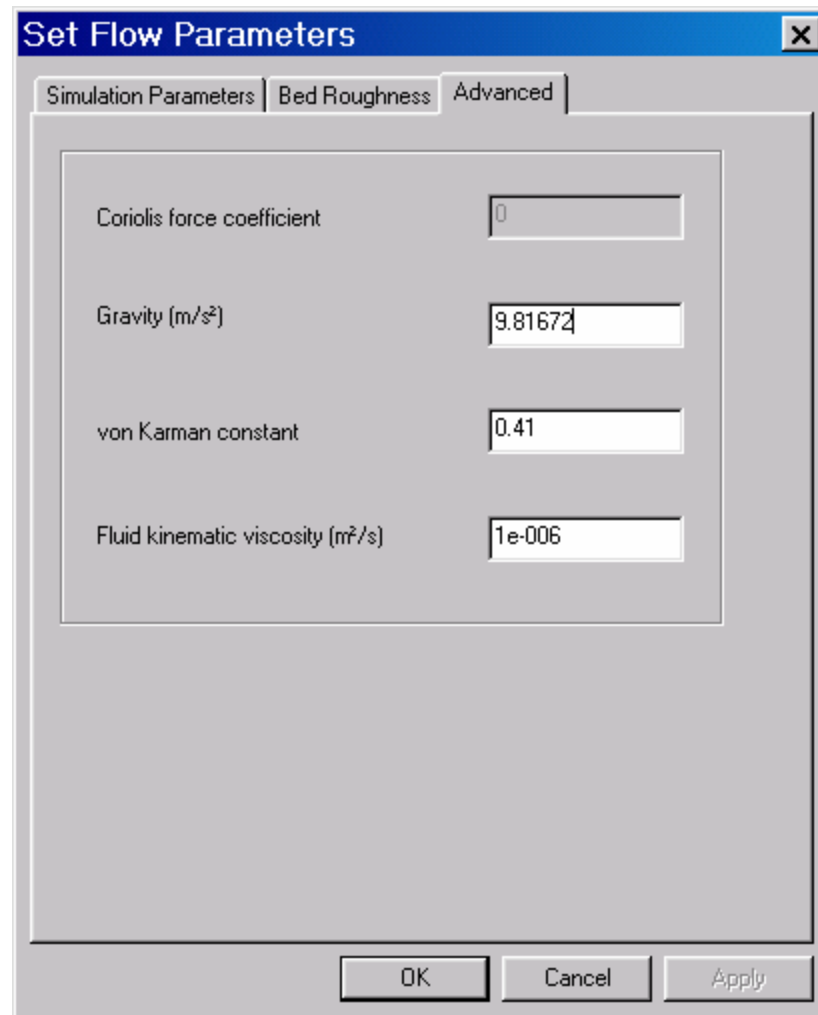





Figure 2-11

- Click **OK** to save the flow parameters you specified.

2.5 Set Inlet and Outlet Boundary Conditions

Initially all the boundaries of the domain are assumed as walls. To set inlet and outlet boundary conditions, there are two steps to follow: editing boundary node strings and attaching boundary conditions to the node strings.

- Select **Start Editing Inlet/Outlet Boundary** from menu **Simulation** or button  to activate the editing toolbar .
- First we set the inlet boundary conditions.
 - Click  and then click the start and the ending points along the first mesh J line. In **Select Inlet/Outlet Boundary** dialog, choose option **Inlet Boundary Condition** and then click **OK**.

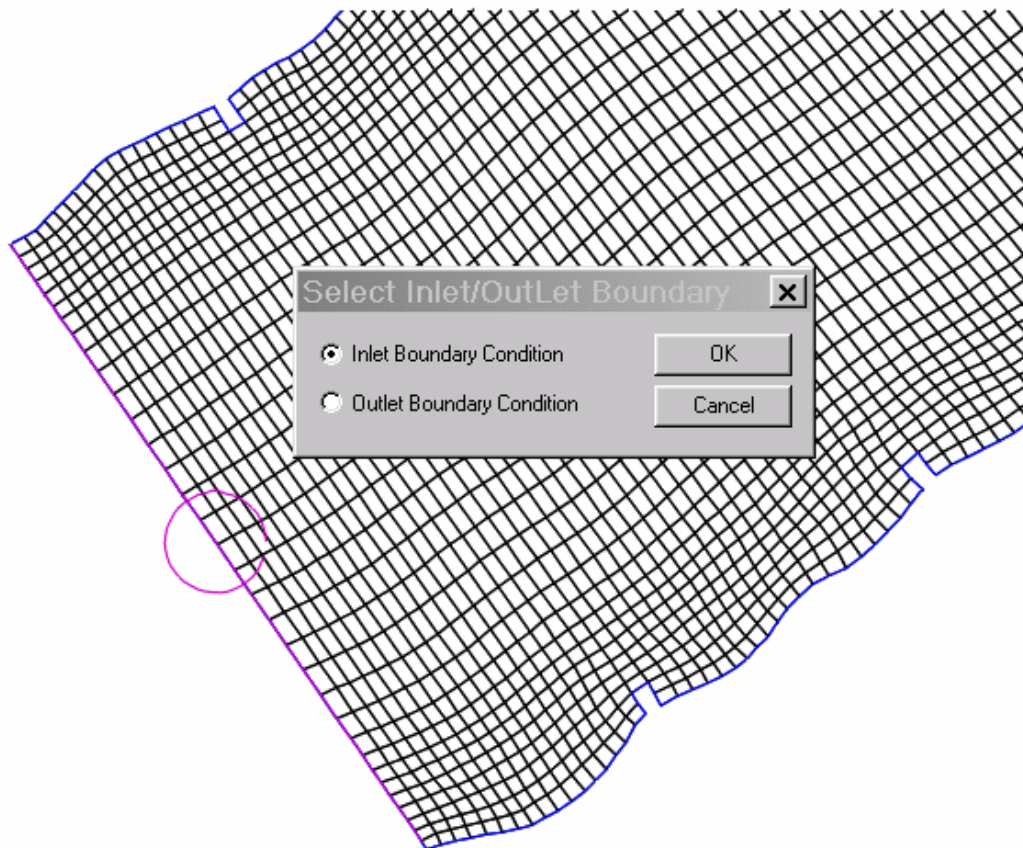


Figure 2-12

- In **Inlet Boundary Conditions** dialog,
 - In the **Flow** page, select option **Total discharge** and enter the value 570.0.
 - Click **OK** to save the inlet boundary conditions.

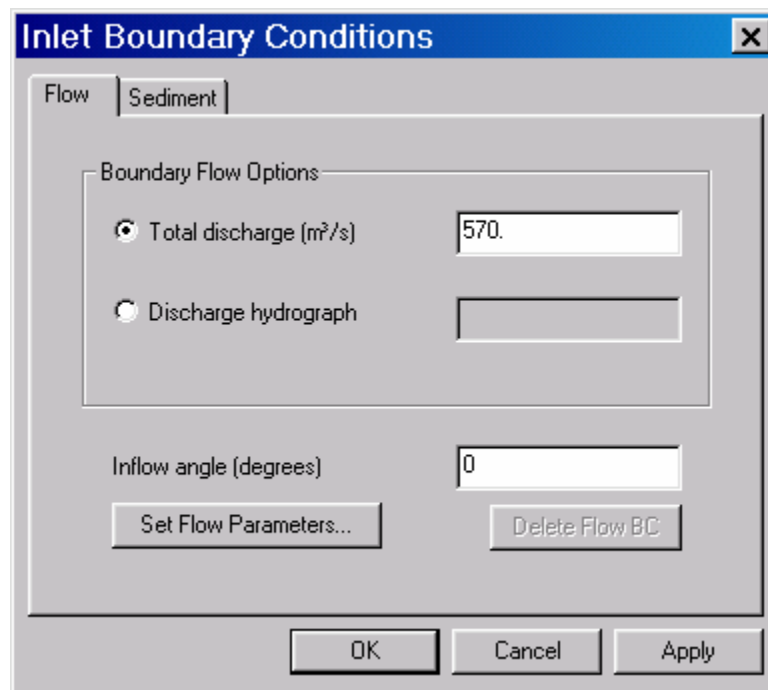



Figure 2-13

- Then we set the outlet boundary conditions.
 - Click  and then click the start and the ending points along the last mesh J line. In **Select Inlet/Outlet Boundary** dialog, choose option **Outlet Boundary Condition** and then click **OK**.

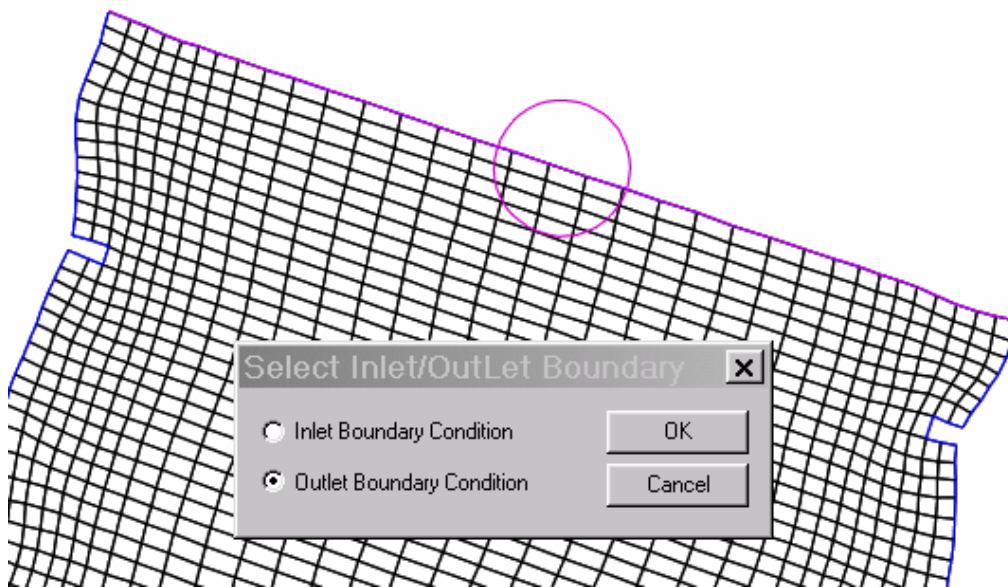


Figure 2-14

- In **Outlet Flow Boundary Condition** dialog, select option **Water surface level**, and then enter the value of 10. Finally click **OK** to save the outlet boundary condition.

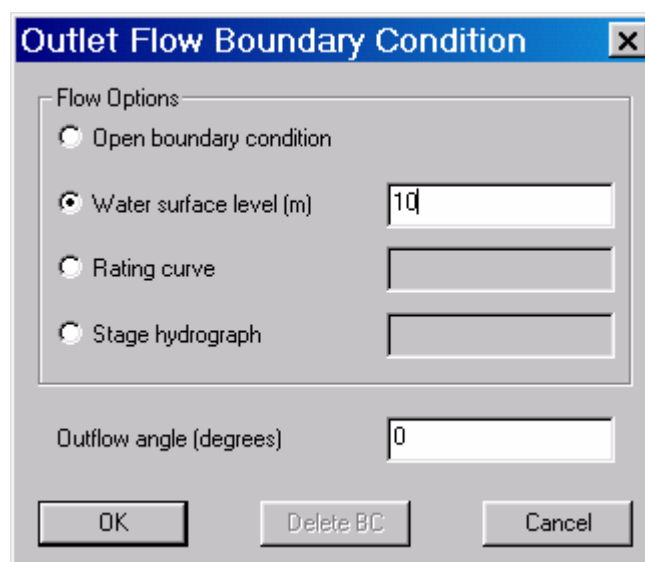



Figure 2-15

2.6 Run CCHE2D Model

After all the initial conditions and the boundary conditions are set, the simulation can be performed.

- Select **Run CCHE2D Model...** in menu **Simulation** or click  in the main toolbar to activate the **Simulation Options** dialog window.

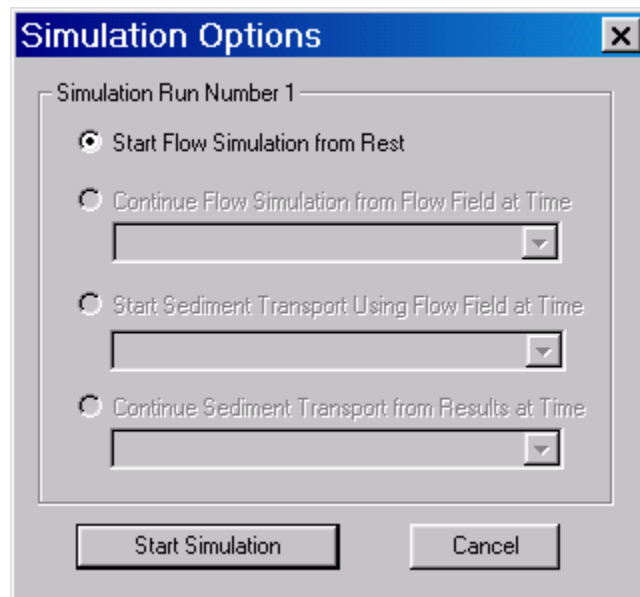
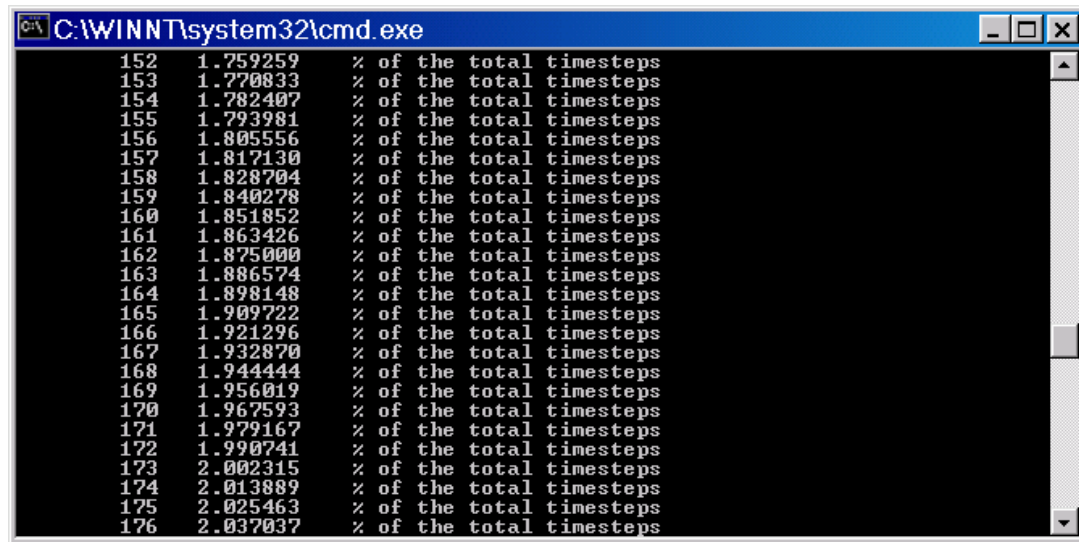


Figure 2-16

- Select option **Start Flow Simulation from Rest** (only this option is available for the first-time run), and then click **Start Simulation**. The flow simulation will be carried out in a console window.



```
C:\WINNT\system32\cmd.exe
152 1.759259 % of the total timesteps
153 1.770833 % of the total timesteps
154 1.782407 % of the total timesteps
155 1.793981 % of the total timesteps
156 1.805556 % of the total timesteps
157 1.817130 % of the total timesteps
158 1.828704 % of the total timesteps
159 1.840278 % of the total timesteps
160 1.851852 % of the total timesteps
161 1.863426 % of the total timesteps
162 1.875000 % of the total timesteps
163 1.886574 % of the total timesteps
164 1.898148 % of the total timesteps
165 1.909722 % of the total timesteps
166 1.921296 % of the total timesteps
167 1.932870 % of the total timesteps
168 1.944444 % of the total timesteps
169 1.956019 % of the total timesteps
170 1.967593 % of the total timesteps
171 1.979167 % of the total timesteps
172 1.990741 % of the total timesteps
173 2.002315 % of the total timesteps
174 2.013889 % of the total timesteps
175 2.025463 % of the total timesteps
176 2.037037 % of the total timesteps
```

Figure 2-17

2.7 Visualize Flow Results

After you start simulations and the intermediate results are available, you can visualize the results. For flow, there are three kinds of result files available, flow intermediate file (*.mdw), flow final results file (*.flw), and flow history file (*.his).

- During the simulation, you can visualize the intermediate file any time.
 - To load it manually, select **Flow Intermediate File** in **Visualization** menu.

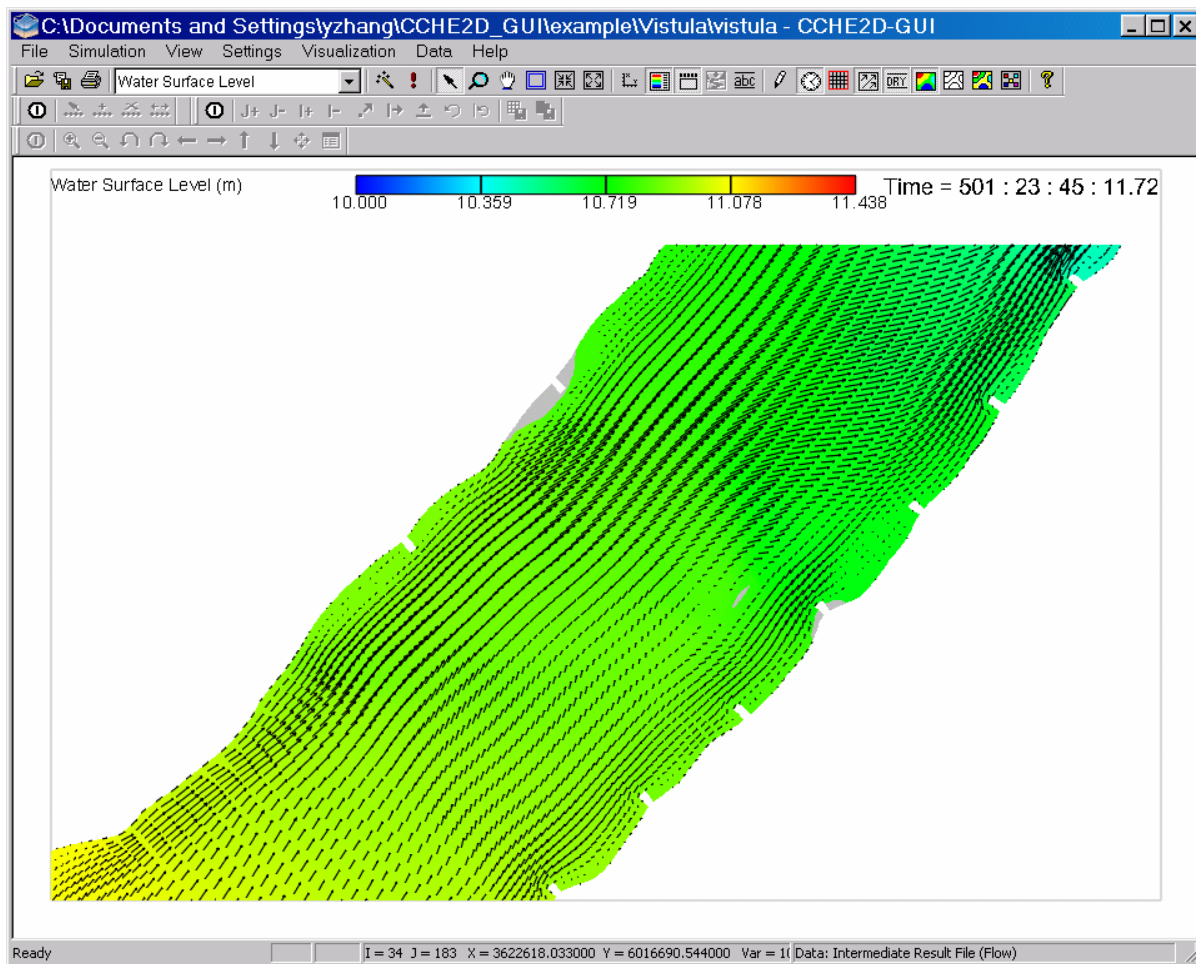


Figure 2-18

- To load it automatically, select **Auto-check Flow Intermediate Result**. In **Set Time Interval** dialog, set the time interval as 10 and then click **OK**.

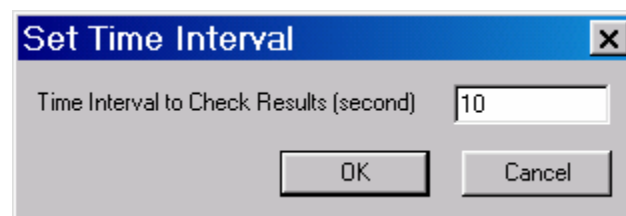


Figure 2-19

- After the simulation is finished, you can visualize the final results file.
 - Select **Flow Final Results File** in **Visualization** menu.
 - In **Select Flow Results File** dialog window, select the flow field and then click **OK**.

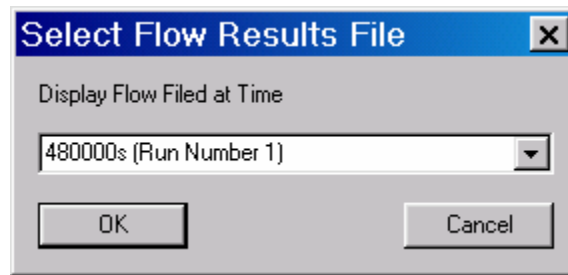


Figure 2-20

- After a result file (**mdw** or **flw**) is loaded,
 - You can select flow variables from **variable selector** on the main toolbar.

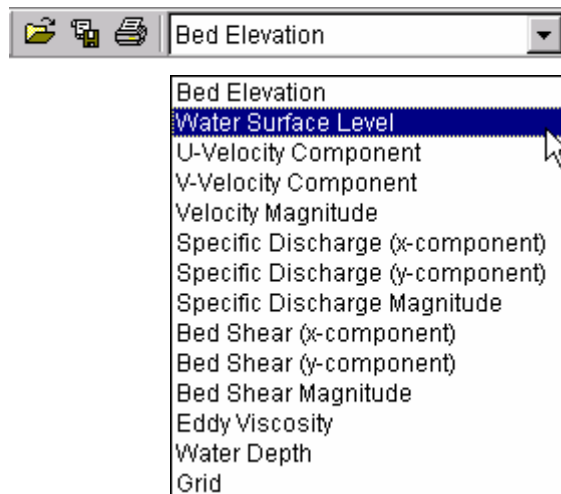





Figure 2-21

- You can personalize your view styles through **Settings** menu. You can show or hide the **Legend** , **Title** , **Grid** , **Boundary**, **Simulation Time**

 , **Dry Area**  , **Frame**, **XY Axis**  and **Velocity Vector**  , and add **Text** .

- In **Settings** menu, you can also select **Color Scale**, **Gray Color** or **Reverse Color Map**, set **Background Color** and choose **Plot Types**.

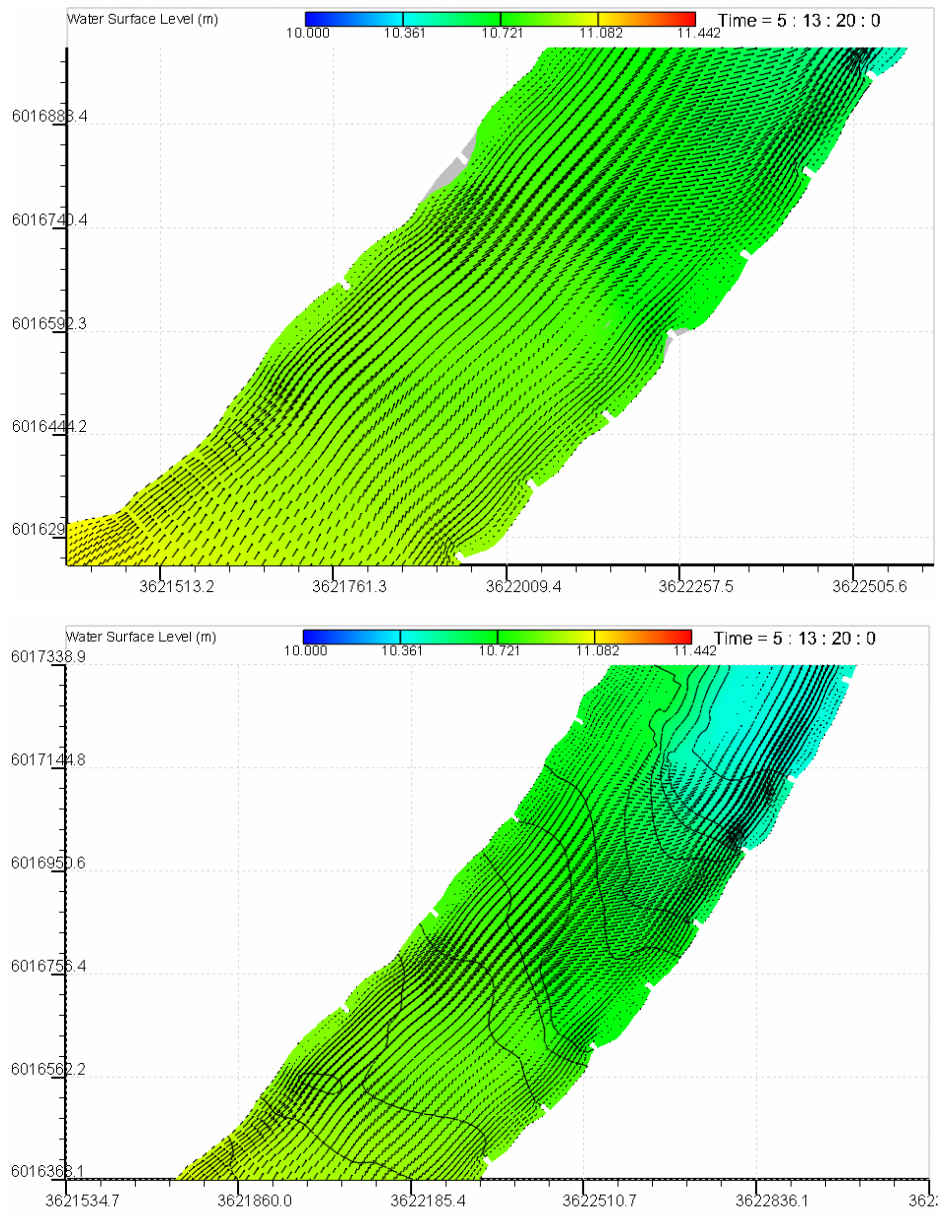


Figure 2-22

- After the simulation is finished, you can use the **History File Editor** to manipulate the visualization of the history results.
 - To load the history file, select **Flow History File** in **Visualization** menu. The **History File Editor** will appear.

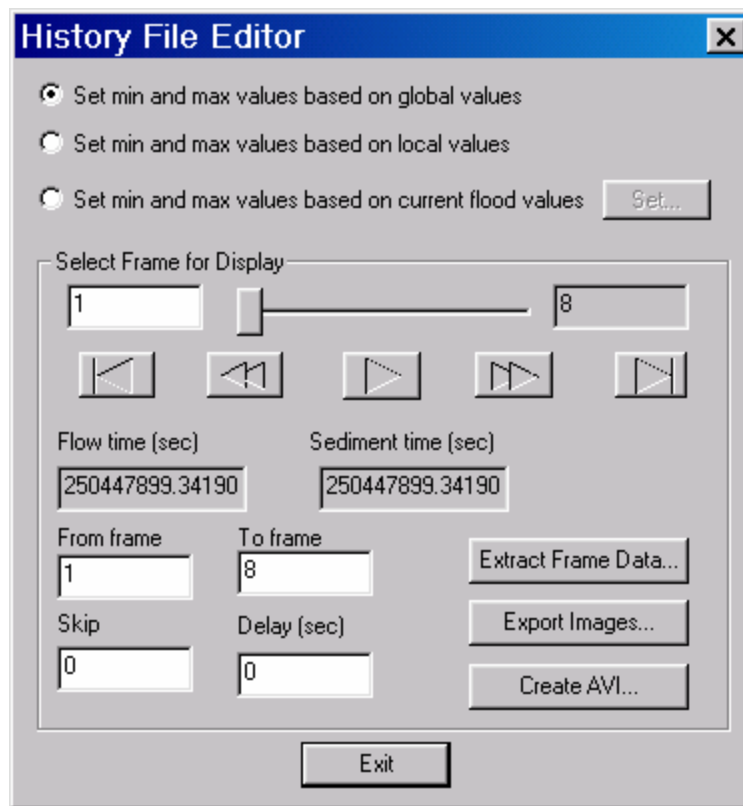



Figure 2-23

- You can **play** () the frames, **Extract Frame Data** to a **flw** file, **Export** frames as Bitmap **Images**, and **Create AVI** for frames.
- Click **Extract Frame Data...** and save the frame data into a **.flw** file. The name of the flw file has special rules (for details, please refer to Chapter 4 section 4.11 of CCHE2D-GUI Users' Manual). Here let's put a name of "**Vistula_Run-3(0).flw**" and then click **Save**. This flw file can be used as start flow field for later runs.

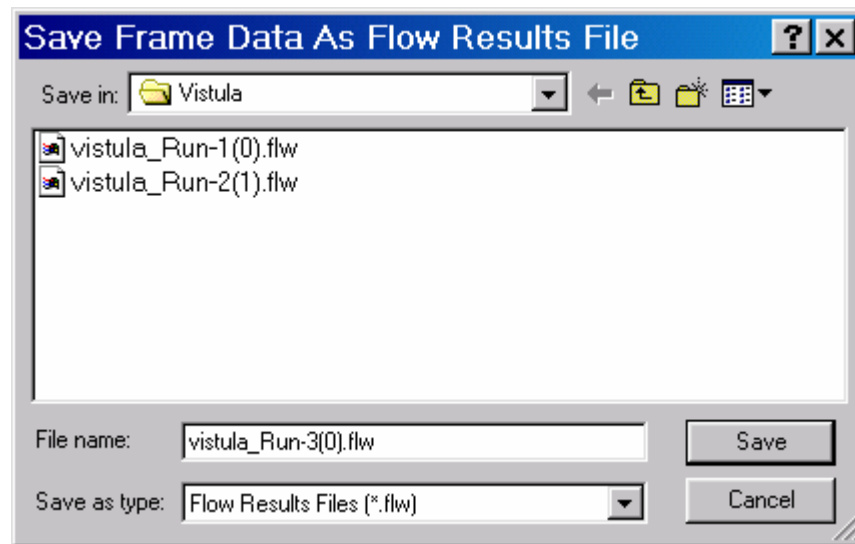


Figure 2-24

- Click **Export Images...** and the selected frames will be saved into Bitmap images.

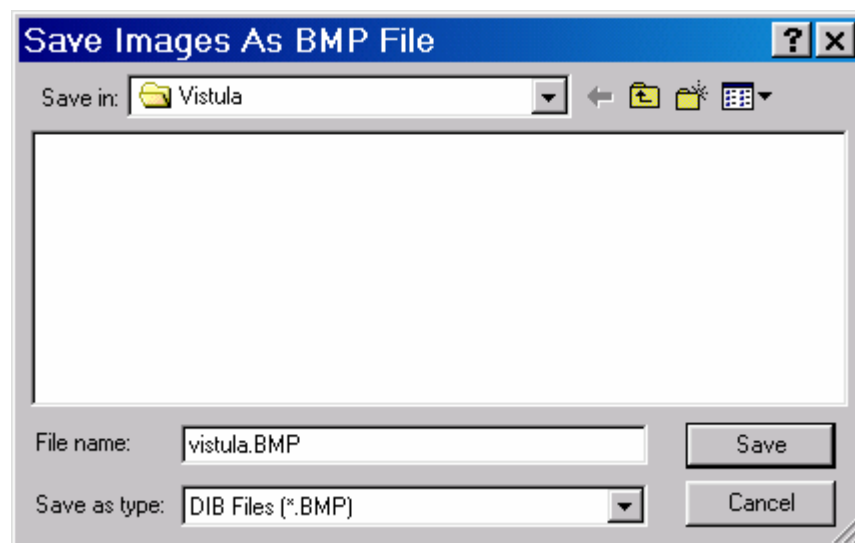


Figure 2-25

- Click **Create AVI...** and the selected frames will be saved into a AVI movie file.

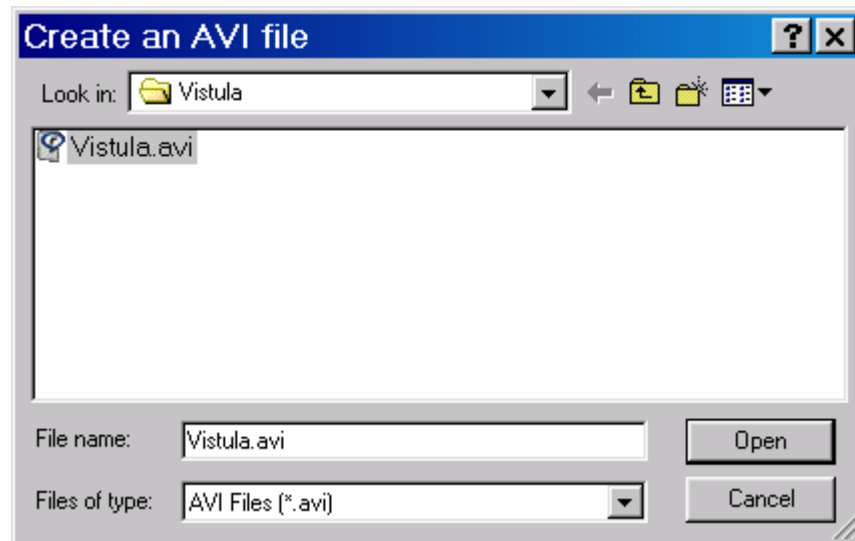


Figure 2-26


3 Example 2: Steady Flow Simulation with Sediment Transport

3.1 Introduction

In Chapter 2, the steady flow simulation of the Vistula River has been conducted. This chapter will illustrate how to use CCHE2D-GUI to simulate sediment transport for the Vistula River. All data is included as example file in the installation package of CCHE2D-GUI.

3.2 Open Geo File

To conduct a simulation, the first step is always to open a geo file.

- Select **Open Geometry...** in **File** menu or click  in the main toolbar. Navigate to the directory of **Vistula** and select the **Vistula.geo**, then click **Open**.

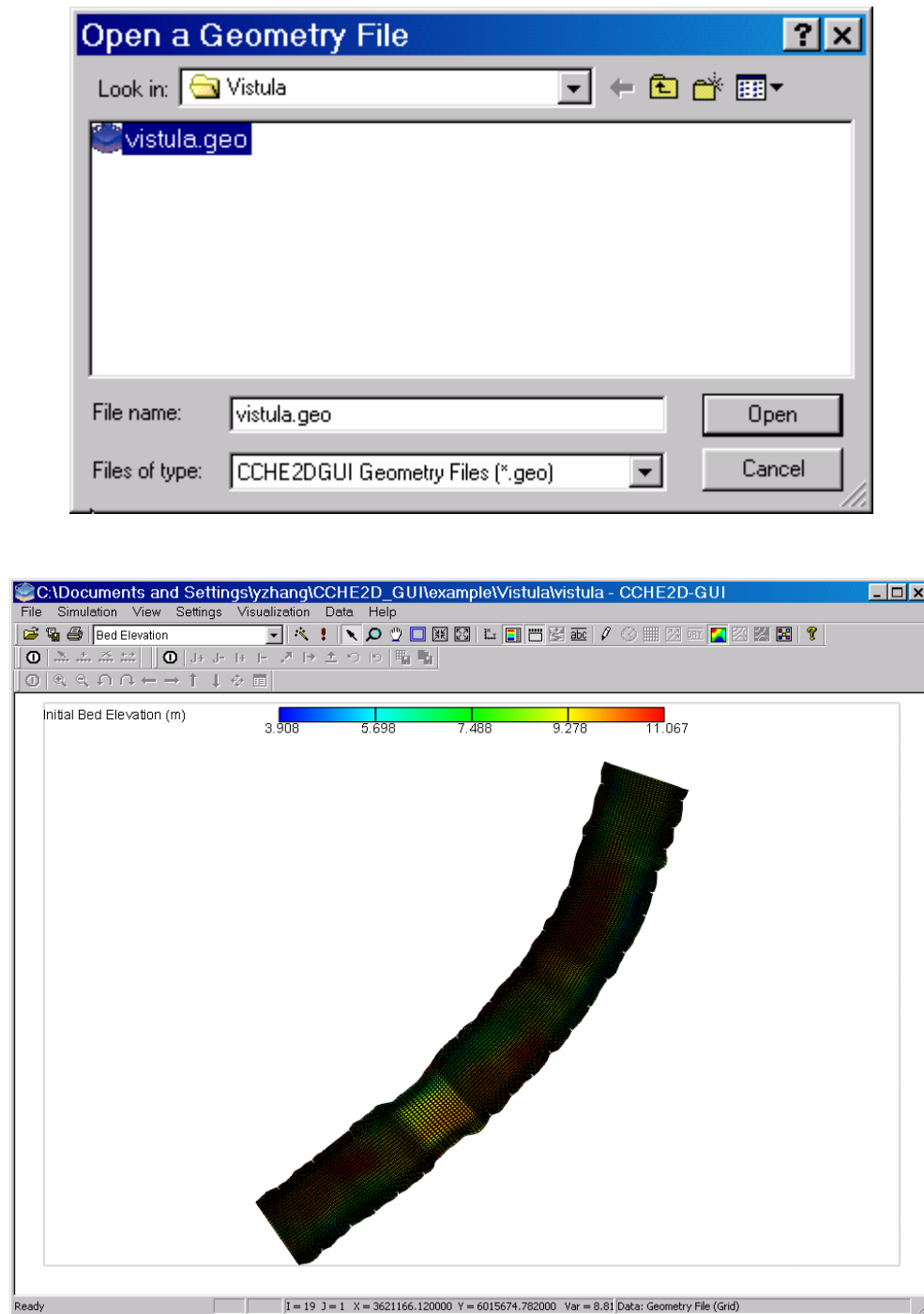


Figure 3-1

3.3 Set Flow Parameters

The flow parameters are already set in Chapter 2 (please refer to section 2.4 for details). We don't need to set them here.

3.4 Set Sediment Parameters

Sediment parameters are specified through a dialog accessible from the **Set Sediment Parameters...** option in the **Simulations** menu. There are four pages, **Sediment Size Classes**, **Sediment Transport**, **Sediment**, and **Bed Roughness**.

- In **Sediment Size Classes** page,
 - First define the Number of Bed Layers as 3 and set the Minimum Mixing Layer Thickness as 0.05
 - Then define the following 7 size classes as shown in Figure 2-12. Each size class is defined using the same procedure: first enter the **Diameter** and then click **Add Size Class**.

The screenshot shows a software window titled "Set Sediment Parameters" with a close button (X) in the top right corner. It has four tabs: "Sediment Size Classes", "Sediment Transport", "Sediment", and "Bed Roughness". The "Sediment Transport" tab is selected. Inside this tab, there are two input fields: "Number of Bed Layers" with the value "3" and "Minimum Mixing Layer Thickness" with the value "0.05". Below these is a section titled "Define Size Class" which contains a list box labeled "Mean diameter (m) of each size class" with a scroll bar. The list contains the following values: 0.00007500, 0.00010000, 0.00015000, 0.00025000, 0.00050000, 0.00100000, and 0.00200000. To the right of the list box is a text input field labeled "Diameter (m)" containing the value "0.002". Below the input field are three buttons: "Add Size Class", "Delete Size Class", and "Clear All". At the bottom of the dialog box are three buttons: "OK", "Cancel", and "Apply".

Figure 3-2

- In **Sediment Transport** page, select **Total Load as Bed Load Plus Suspended Load** transport mode; set the Sediment Simulation Mode as **Slow Bed Change with Steady Flow**; specify the adaptation length for bed load as 100; set the adaptation factor for suspended load as 0.04.

The screenshot shows a software dialog box titled "Set Sediment Parameters" with a close button (X) in the top right corner. The dialog has four tabs: "Sediment Size Classes", "Sediment Transport" (which is selected), "Sediment", and "Bed Roughness".

Under the "Sediment Transport" tab, the following settings are visible:

- Transport Mode:** A dropdown menu set to "Total Load as Bed Load Plus Suspended Load".
- Transport Capacity Formula:** A dropdown menu set to "Wu et al. formula".
- Sediment Simulation Mode:** Two radio buttons. "Slow Bed Change with Steady Flow" is selected, and "Fast Bed Change with Unsteady Flow" is unselected.
- Adaptation Length for Bedload:** Three radio buttons. "Specify adaptation length" is selected, with a text input field next to it containing the value "1000". The other two options, "Set as average grid length" and "Set as 7.3 of average dune length", are unselected.
- Adaptation Factor for Suspended Load:** Two radio buttons. "Specify adaptation factor" is selected, with a text input field next to it containing the value "0.25". The other option, "Based on Armanini and di Silvio (1988)", is unselected.

At the bottom of the dialog are three buttons: "OK", "Cancel", and "Apply".

Figure 3-3

- In Sediment page, set the **Time steps to adjust flow** as 20 and the **Erosion/Deposition limit** as 0.01. We use the default value of the **Sediment specific gravity** and choose not to **include curvature effects**.

The screenshot shows a Windows-style dialog box titled "Set Sediment Parameters". It has four tabs: "Sediment Size Classes", "Sediment Transport", "Sediment", and "Bed Roughness". The "Sediment" tab is currently active. Inside the dialog, there are three main sections. The first section, "Sediment specific gravity", has a text box containing the value "2.65". The second section, "Curvature Effects", contains a checkbox labeled "Include curvature effects" which is unchecked, and a text box for "Average channel width (m)" containing the value "1000". The third section, "Steady Flow Computation", contains a text box for "Time steps to adjust flow" with the value "20", and a text box for "Erosion/Deposition limit (0.01-0.05 of depth)" with the value "0.01". At the bottom of the dialog are three buttons: "OK", "Cancel", and "Apply".

Figure 3-4

- The **Bed Roughness** page in **Set Sediment Parameters** contains the same parameters in **Set Flow Parameters**. Since we already set the parameters of bed roughness in **Set Flow Parameters**, we don't need to specify them here.
- Click **OK** or **Apply** to save your settings.

3.5 Set Sediment Boundary Conditions

The information of sediment boundary conditions are provided by the suspended sediment boundary condition file (*.sbc) and the bedload boundary condition file (*.bbc). The CCHE2D-GUI provides a file editor to edit these two files.

- Select **Set Sediment Boundary Conditions...** in menu **Simulation** to invoke the **File Editor**.

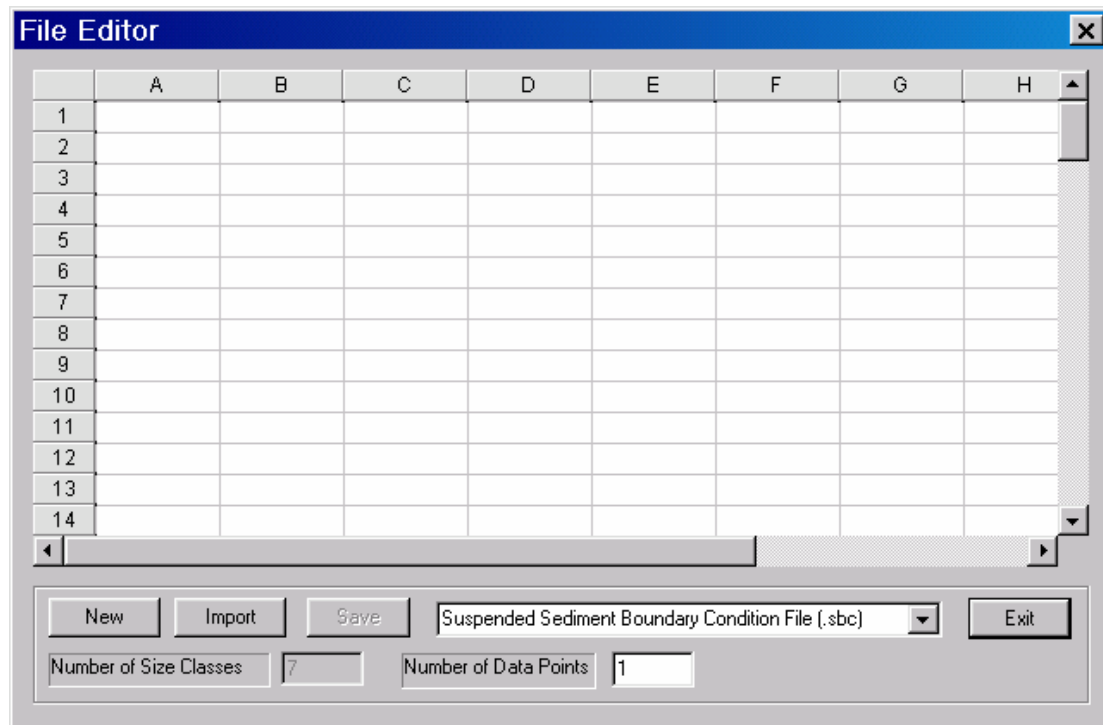


Figure 3-5

- Select **Suspended Sediment Boundary Condition File (.sbc)** from the file selector.
 - Specify the **Number of Data Points** as 1, and then click **New** to create a template for sbc file.

The File Editor window displays a table with 14 rows and 8 columns (A-H). The first three rows contain headers and units, and the first row of data contains numerical values. The remaining rows are empty.

	A	B	C	D	E	F	G	H
1	Time	Discharge	Class1	Class2	Class3	Class4	Class5	Class6
2	(s)	(kg/m ³)	(7.5e-005)	(0.0001)	(0.00015)	(0.00025)	(0.0005)	(0.001)
3	0	1	0.142857	0.142857	0.142857	0.142857	0.142857	0.142857
4								
5								
6								
7								
8								
9								
10								
11								
12								
13								
14								

At the bottom of the window, there are buttons for 'New', 'Import', 'Save', and 'Exit'. A dropdown menu shows 'Suspended Sediment Boundary Condition File (.sbc)'. Below these are input fields for 'Number of Size Classes' (set to 7) and 'Number of Data Points' (set to 1).

Figure 3-6

- Enter the desired values for **time**, **discharge** and the **fractions** of each size class in the corresponding cells. The fractions should be 0.4, 0.2, 0.2, 0.2, 0.0, 0.0 and 0.0.

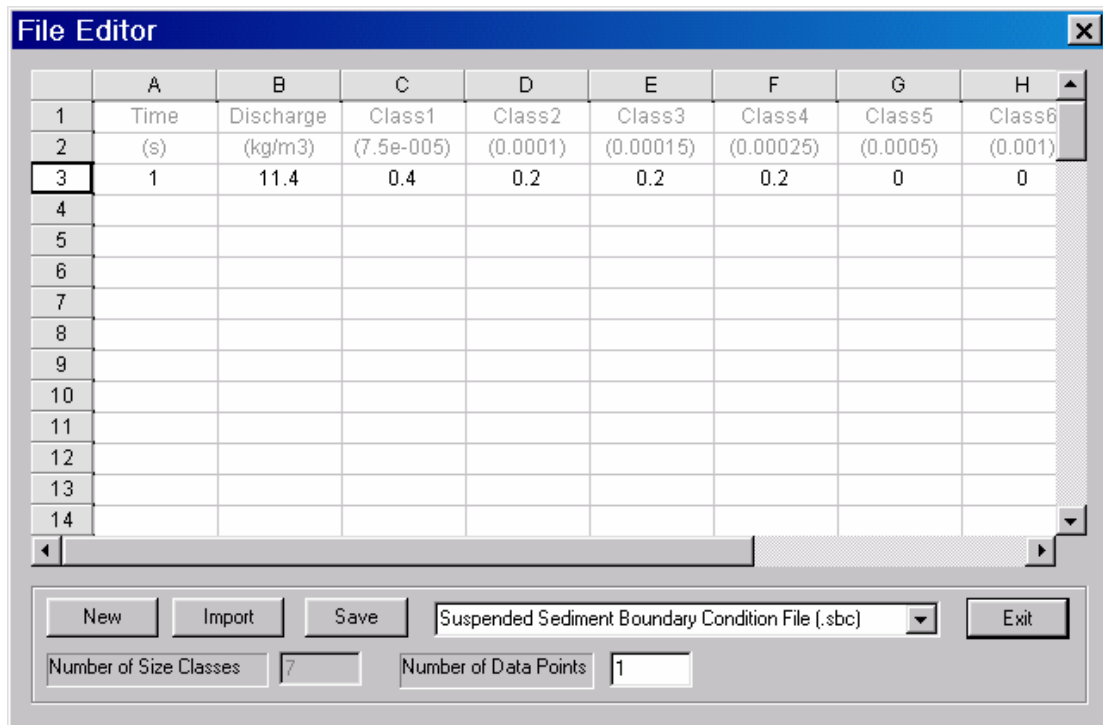


Figure 3-7

- Click **Save** to save your changes into file Vistula.sbc.

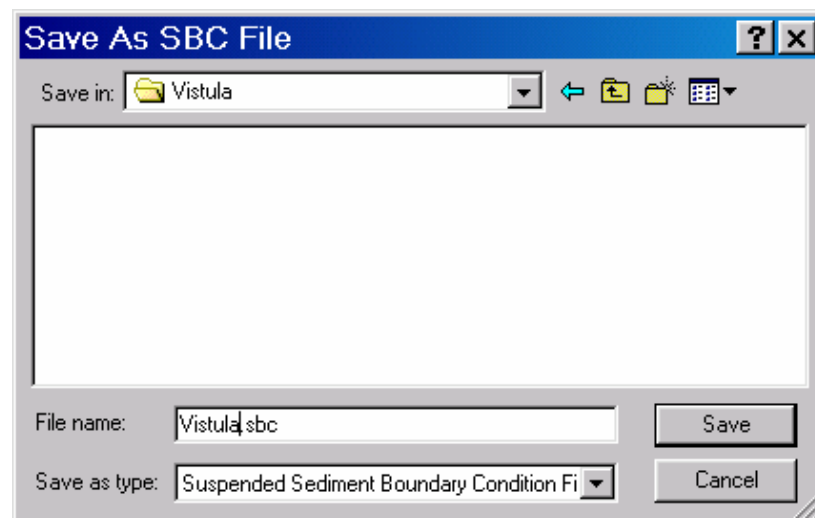


Figure 3-8

- Select **BedLoad Boundary Condition File (.bbc)** from the file selector.
 - Specify the **Number of Data Points** as 1, and then click **New** to create a template for bbc file.

	A	B	C	D	E	F	G	H
1	Time	Discharge	Class1	Class2	Class3	Class4	Class5	Class6
2	(s)	(kg/s)	(7.5e-005)	(0.0001)	(0.00015)	(0.00025)	(0.0005)	(0.001)
3	0	1	0.142857	0.142857	0.142857	0.142857	0.142857	0.142857
4								
5								
6								
7								
8								
9								
10								
11								
12								
13								
14								

New Import Save BedLoad Boundary Condition File (.bbc) Exit
 Number of Size Classes 7 Number of Data Points 1

Figure 3-9

- Enter the desired values for **time**, **discharge** and the **fractions** of each size class in the corresponding cells. The fractions should be 0.054, 0.06895, 0.26835, 0.5465, 0.0459, 0.01485 and 0.00145.

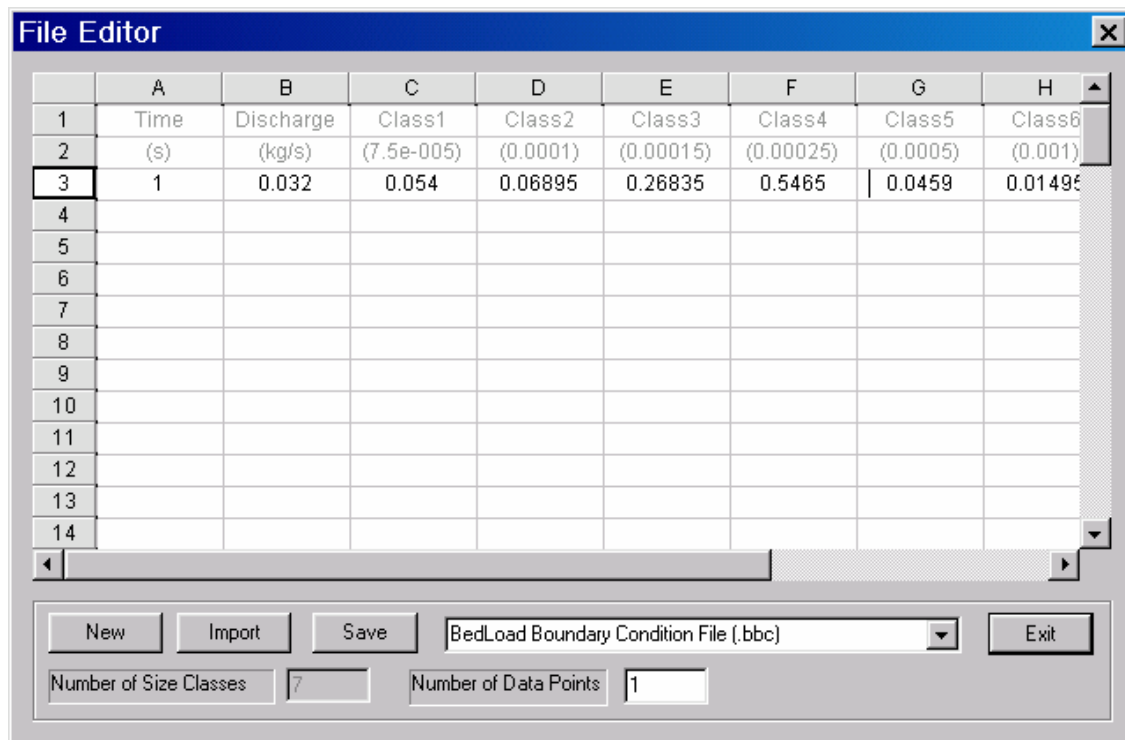


Figure 3-10

- Click **Save** to save your changes into file Vistula.bbc.

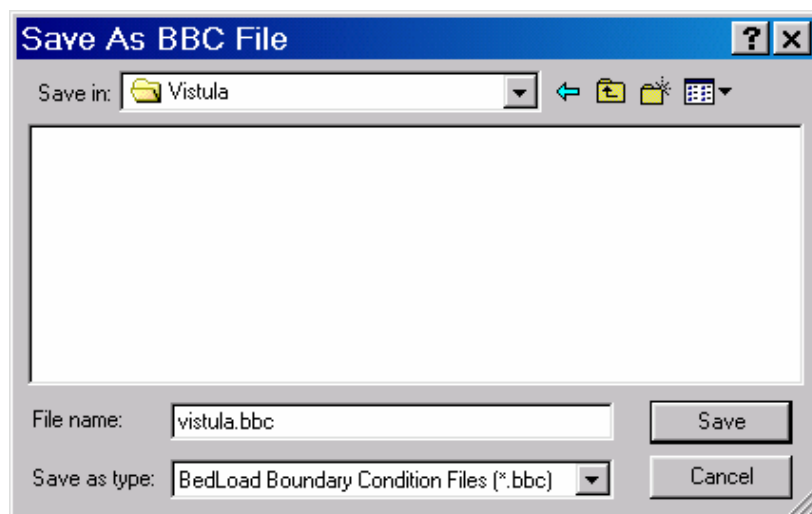





Figure 3-11

3.6 Set Inlet and Outlet Boundary Conditions

In Chapter 2, we already set the flow boundary conditions of the inlet and outlet. In this chapter, we only need to set the sediment inlet boundary conditions.

- Select **Start Editing Inlet/Outlet Boundary** from menu **Simulation** or button  to activate the editing toolbar .
- Click  and then select the inlet node string which is indicated by an arrow pointed into the domain. The **Inlet Boundary Conditions** dialog will be activated.

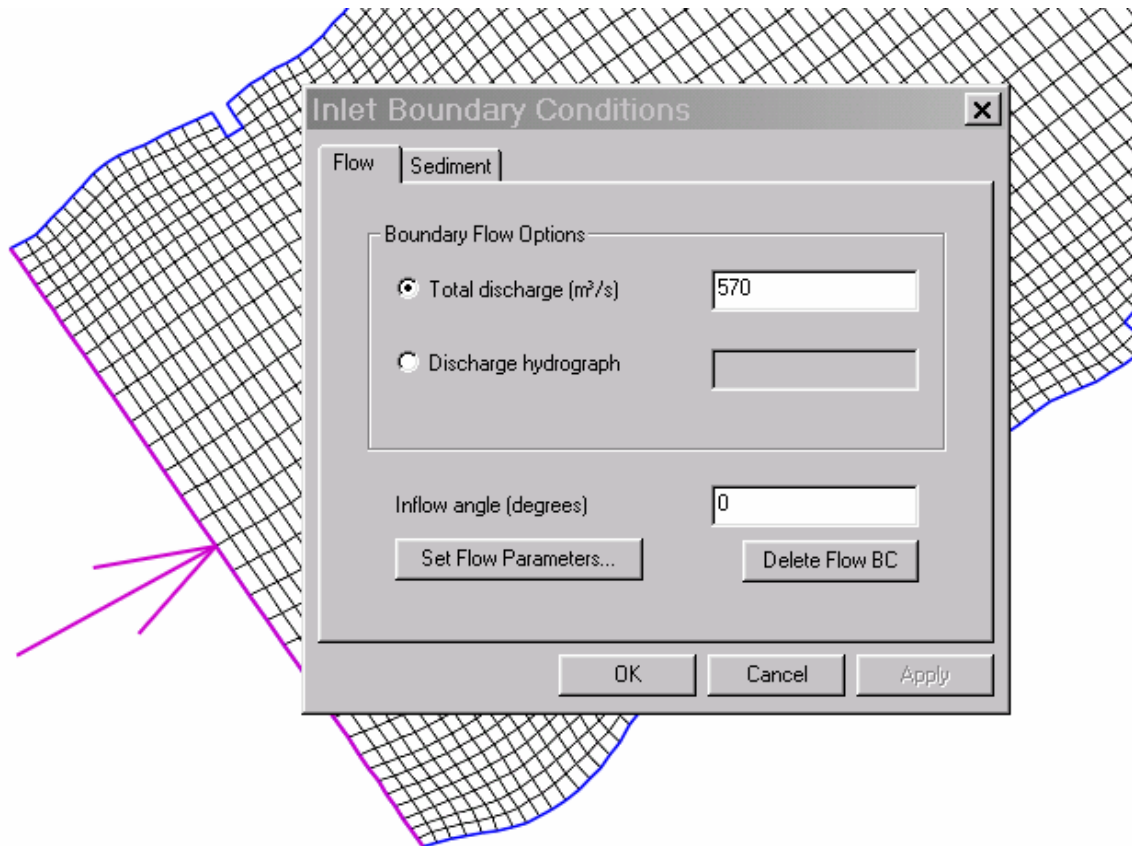
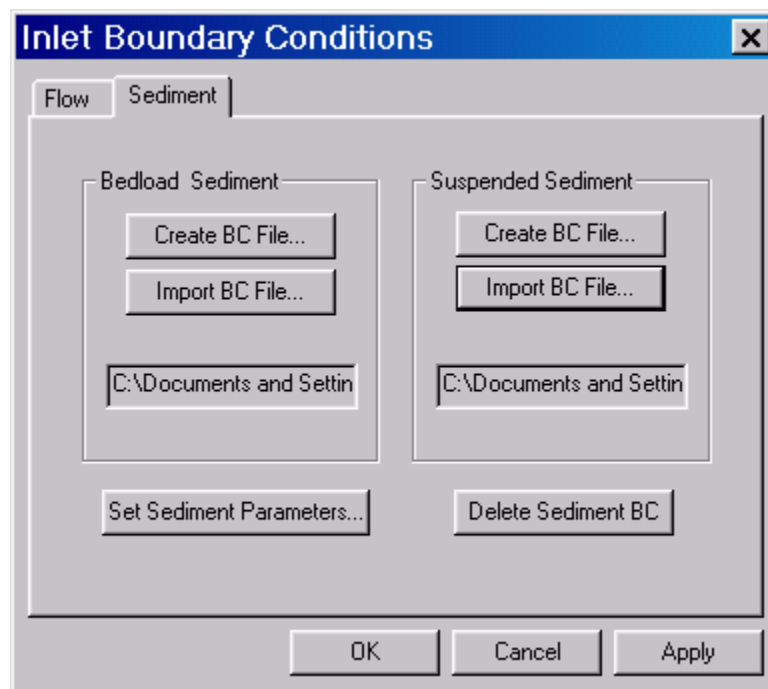


Figure 3-12

- In **Inlet Boundary Conditions** dialog,

- In page **Sediment**, click **Import BC File...** in the group **BedLoad Sediment**, and in **Open a Bedload Sediment File** dialog choose **vistula.bbc** and then click **Open**.
- Click **Import BC File...** in the group **Suspended Sediment**, and in **Open a Suspended Sediment File** dialog choose **vistula.sbc** and then click **Open**.
- Click **OK** to save the inlet boundary conditions.



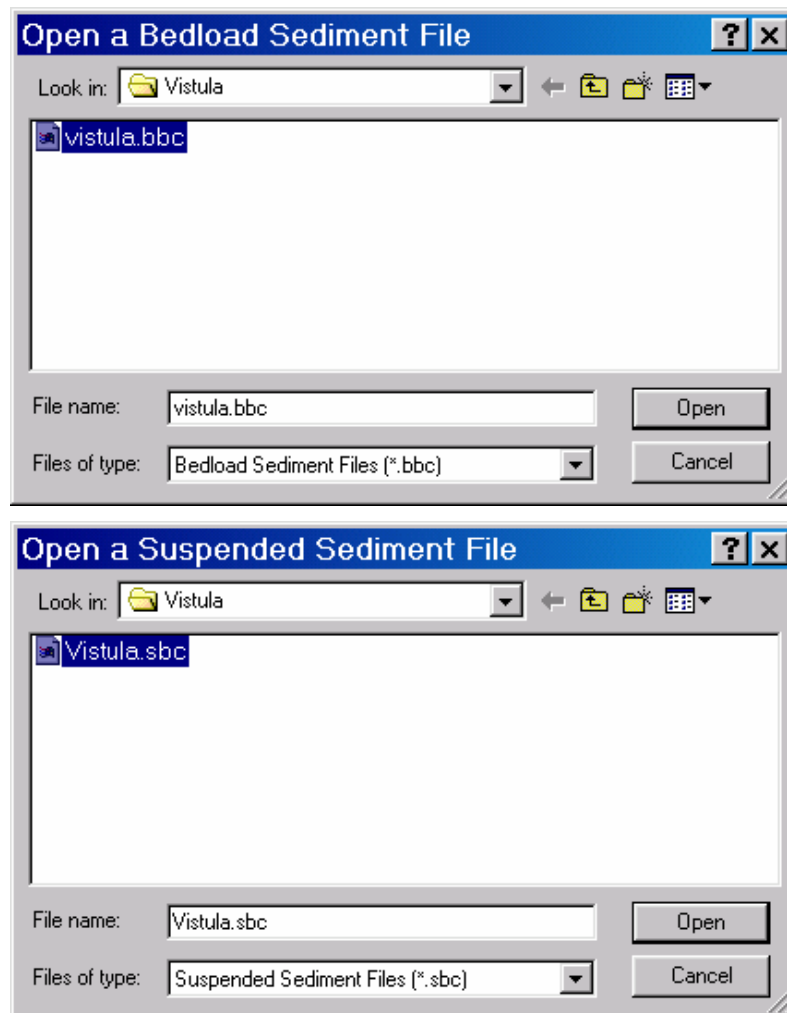


Figure 3-13

3.7 Set Bed Material Samples

The bed material samples will be used to define the initial bed material compositions in both horizontal and vertical directions for the entire domain. The information of the bed material samples are contained in a file with the extension **bmt**. You may create as many samples as necessary. In this example, however, we assume the same sediment size distribution throughout the domain.

- First select **Set Bed Material Samples...** in menu **Simulation** to activate **Define Bed Material Samples** dialog window.

	A	B	C	D	E	F	G
1	Sample No.	Porosity	Class 1	Class 2	Class 3	Class 4	Class 5
2			(7.5e-005)	(0.0001)	(0.00015)	(0.00025)	(0.0005)
3							
4							
5							
6							
7							
8							
9							

Figure 3-14

- In **Define Bed Material Samples** dialog,
 - Click **Add Sample** to create a new template of a sample which has the uniform size distribution and the default porosity of 0.24.

	A	B	C	D	E	F	G
1	Sample No.	Porosity	Class 1	Class 2	Class 3	Class 4	Class 5
2			(7.5e-005)	(0.0001)	(0.00015)	(0.00025)	(0.0005)
3	1	0.24	0.142857	0.142857	0.142857	0.142857	0.142857
4							
5							
6							
7							
8							
9							

Figure 3-15

- Enter the correct values for **porosity** and the **fractions** of each size class in the corresponding cells. The fractions should be 0.054, 0.06895, 0.26835, 0.5465, 0.0459, 0.01485 and 0.00145.

	A	B	C	D	E	F	G
1	Sample No.	Porosity	Class 1	Class 2	Class 3	Class 4	Class 5
2			(7.5e-005)	(0.0001)	(0.00015)	(0.00025)	(0.0005)
3	1	0.3	0.054	0.06895	0.26835	0.5465	0.0459
4							
5							
6							
7							
8							
9							

Buttons: Reset, Add Sample, OK, Cancel

Figure 3-16

- Click **OK** to save the samples.

3.8 Set Bed Material Properties

After the bed samples are defined, we need to set the initial bed material properties contained in a file with the extension **bed**. For each mesh node, we must define a series of properties that are required for the simulation of sediment transport. There are five properties divided into two groups for each mesh node, **Layer Thickness**, **Bed Sample Number**, **Erodibility**, **Maximum Erosion Thickness**, and **Maximum Deposition Thickness**. Some properties, such as **Layer Thickness** and **Bed Sample Number**, must be defined for each layer in the bed.

- Select **Set Bed Material Properties...** in menu **Simulation** to activate **Modify Bed Material Properties** dialog window.

Edit Bed Material Properties

Layer properties

Layer Number: Number 1

☒ Layer Thickness (m): 1

☐ Bed Sample Number: Sample 1

Whole Domain Define Distribution

Please specify both thickness and sample number for EACH LAYER.

Nodal Erodibility and Thickness

☒ Erodibility: YES

☐ Maximum Erosion Thickness (m): 90

☐ Maximum Deposition Thickness (m): 90

Whole Domain Define Distribution

Undo Undo All Save Exit

Figure 3-17

- First specify the **Layer properties**.
 - Select **Number 1** from **Layer Number** selector. The total number of bed layer is 3 (see section 2.5).
 - Select option **Layer Thickness** and enter the value of 0.5, and then click **Whole Domain** in group **Layer properties**.
 - Select option **Bed Sample Number** and select **Sample 1** from sample selector, and then click **Whole Domain**. The total number of bed sample is 1 (see section 2.8).
 - Repeat the above steps until properties for all layers are specified. Set **Layer Thickness** 1.0 and 2.0 for the **Number 2** and **Number 3** layer, and set **Sample 1** for the rest two layers.

The figure displays four screenshots of the "Edit Bed Material Properties" dialog box, arranged in a 2x2 grid. Each dialog box has a title bar with a close button (X) and contains two main sections: "Layer properties" and "Nodal Erodibility and Thickness".

Top Left Dialog:

- Layer properties:** Layer Number is set to "Number 1". "Layer Thickness (m)" is selected with a value of 0.2. "Bed Sample Number" is set to "Sample 1". Buttons "Whole Domain" and "Define Distribution" are present.
- Nodal Erodibility and Thickness:** "Erodibility" is selected and set to "YES". "Maximum Erosion Thickness (m)" and "Maximum Deposition Thickness (m)" are both set to 90. Buttons "Whole Domain" and "Define Distribution" are present.
- Buttons:** "Undo", "Undo All", "Save", and "Exit" are at the bottom.

Top Right Dialog: This dialog is identical to the top left one.

Bottom Left Dialog:

- Layer properties:** Layer Number is set to "Number 2". "Layer Thickness (m)" is selected with a value of 1.0. "Bed Sample Number" is set to "Sample 1". Buttons "Whole Domain" and "Define Distribution" are present.
- Nodal Erodibility and Thickness:** "Erodibility" is selected and set to "YES". "Maximum Erosion Thickness (m)" and "Maximum Deposition Thickness (m)" are both set to 90. Buttons "Whole Domain" and "Define Distribution" are present.
- Buttons:** "Undo", "Undo All", "Save", and "Exit" are at the bottom.

Bottom Right Dialog: This dialog is identical to the bottom left one.

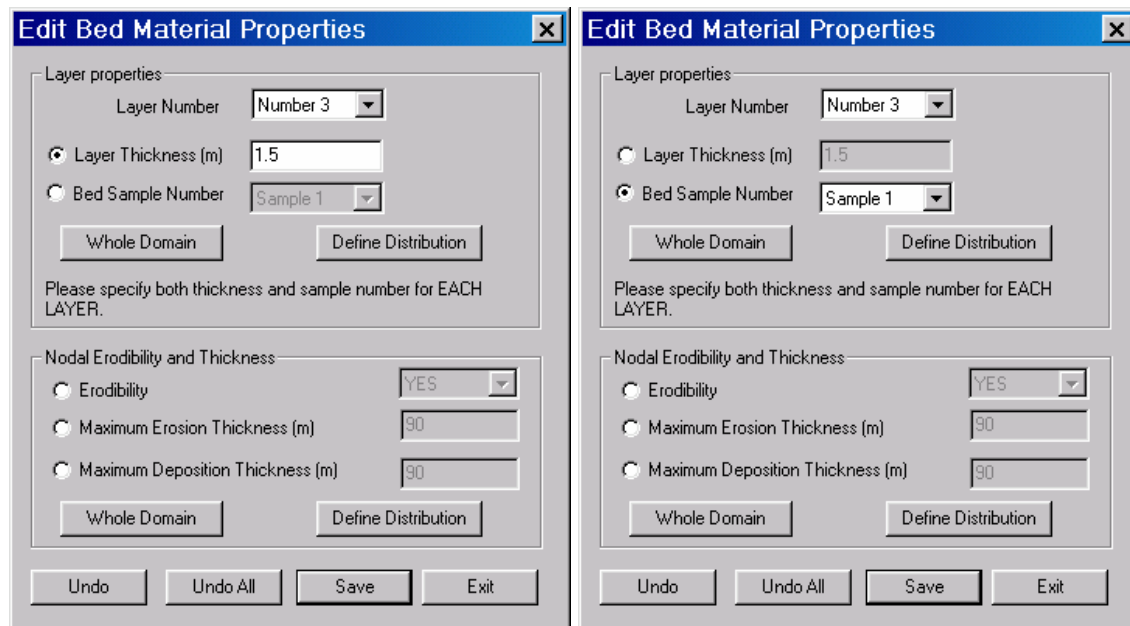


Figure 3-18

- Then let's specify the nodal erodibility and thickness.
 - Select option **Erodibility** and select **YES**, and then click **Whole Domain** in group **Nodal Erodibility and Thickness**. This indicates that the bed is erodible throughout the domain.
 - Select option **Maximum Erosion Thickness** and enter the value of 80, and then click **Whole Domain**.
 - Select option **Maximum Deposition Thickness** and enter the value of 80, and then click **Whole Domain**.
 - Note that we specified very large erosion and deposition thickness to indicate that virtually no limit should

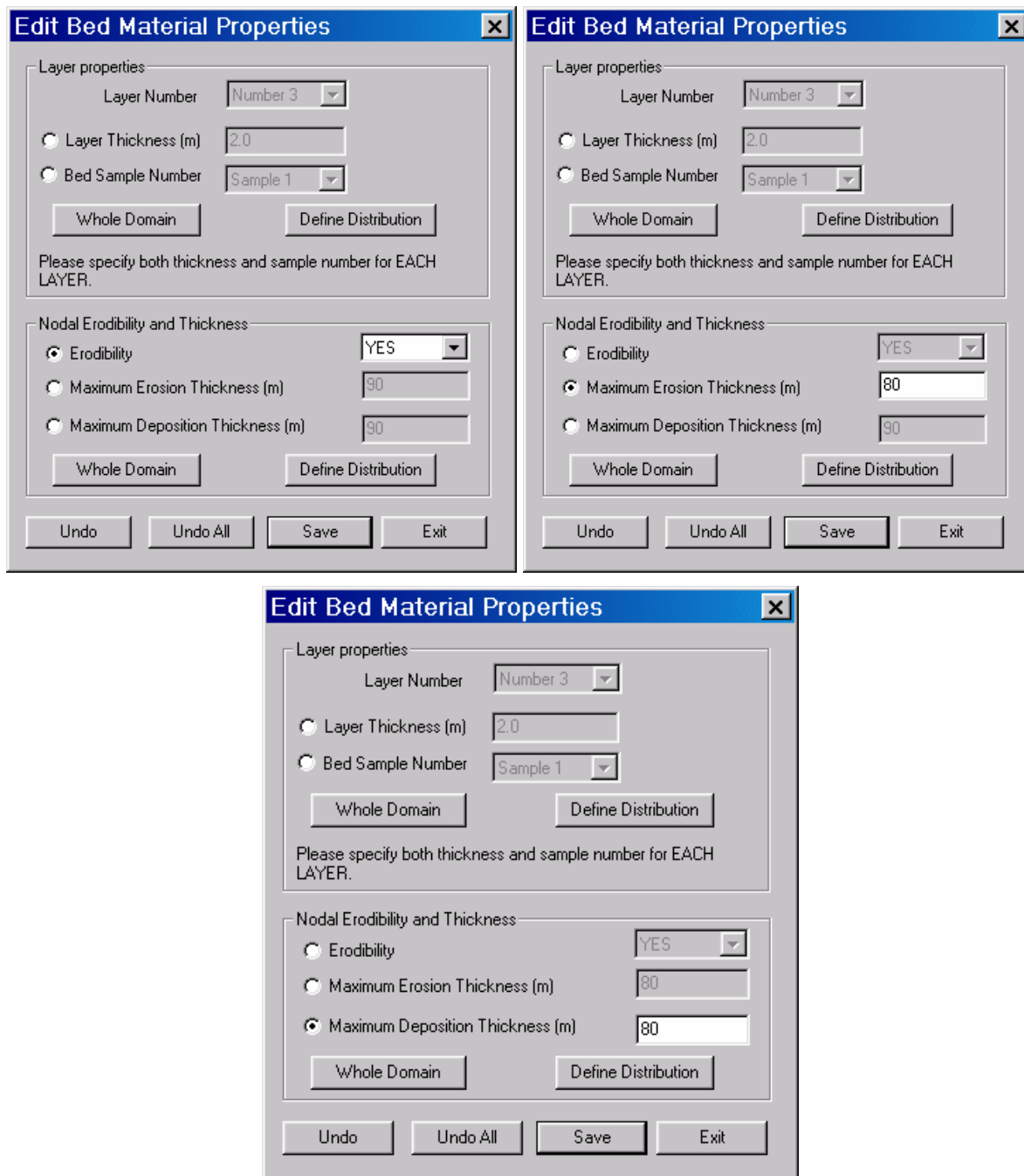



Figure 3-19

- Finally click **Save** to save the changes, and then click **Exit**.

3.9 Run CCHE2D Model

After all the initial conditions and the boundary conditions are set, the simulation can be performed.

- Usually the simulation of sediment transport is divided into two steps. First run steady flow simulation. After flow simulation is finished, we run sediment transport simulation. In Chapter 2, the flow simulation is already finished. We can directly run the sediment transport
- Invoke the **Simulation Options** dialog window by selecting **Run CCHE2D Model ...** in the menu **Simulation** or clicking  in the main tool bar. Then select option **Start Sediment Transport using Flow Field at Time**. There is only one flow field available. Select that flow field and click **Start Simulation**. The sediment transport simulation will begin.

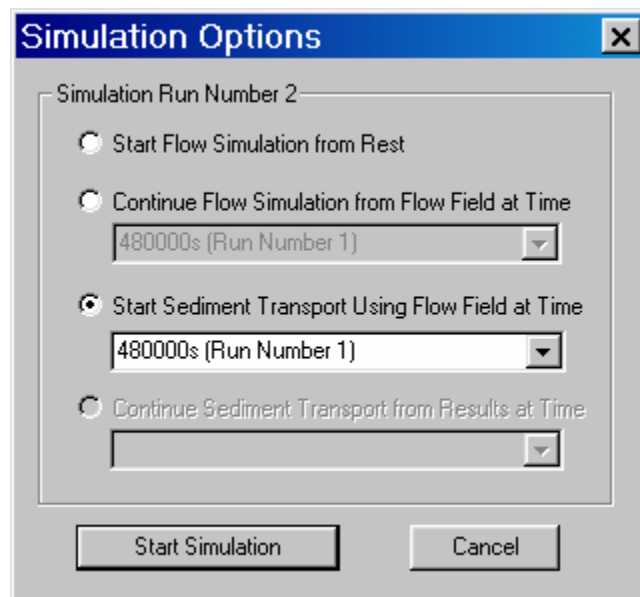


Figure 3-20

```

C:\WINNT\system32\cmd.exe
NUMBER OF DRY NODES AT THE BEGINNING ARE      398
PROGRAM RUNS
No In-stream structure is specified
   1  92.5925925925926      % of the total timesteps
   2  92.5925925925926      % of the total timesteps
   3  92.5925925925926      % of the total timesteps
   4  92.5925925925926      % of the total timesteps
   0  0.0000000000000E+000  5.0000000000000      %
TIMESCALE FOR SEDIMENT ENLARGED
TIME SCALE FOR SEDIMENT  10.0000000000000      1.2500000000000

   5  92.7083333333333      % of the total timesteps
   6  92.7083333333333      % of the total timesteps
   7  92.7083333333333      % of the total timesteps
   8  92.7083333333333      % of the total timesteps
   9  92.7083333333333      % of the total timesteps
   0  0.0000000000000E+000  5.0000000000000      %
TIMESCALE FOR SEDIMENT ENLARGED
TIME SCALE FOR SEDIMENT  10.0000000000000      1.5625000000000

  10  92.8530092592593      % of the total timesteps
  11  92.8530092592593      % of the total timesteps
  12  92.8530092592593      % of the total timesteps
  13  92.8530092592593      % of the total timesteps
  14  92.8530092592593      % of the total timesteps

```

Figure 3-21

3.10 Visualize Sediment Results

Similarly to flow result files, there are also three kinds of sediment result files, sediment intermediate file(.mds), sediment final results file (.sdm), and sediment history file (.sed).

- During the simulation, you can visualize the intermediate file any time.
 - To load it manually, select **Sediment Intermediate File** in **Visualization** menu.

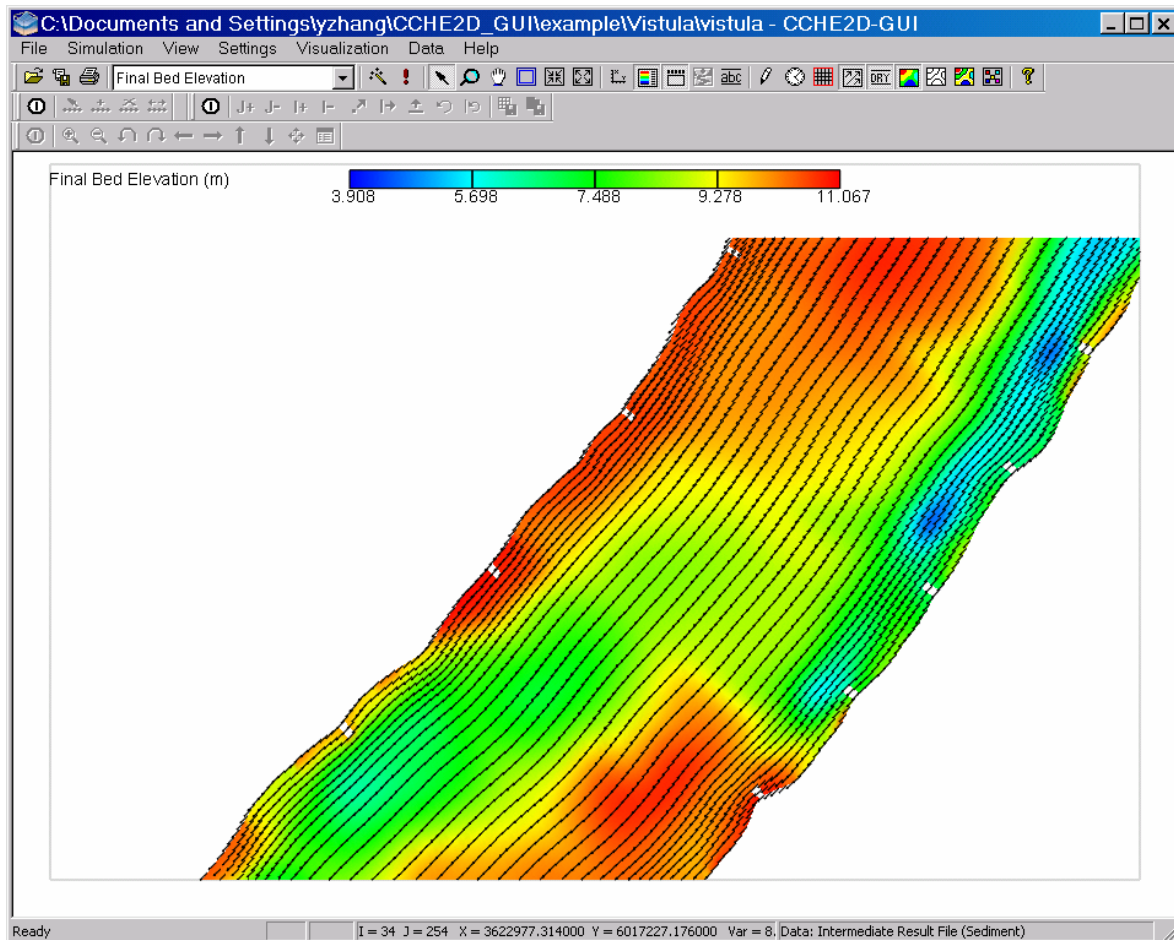


Figure 3-22

- To load it automatically, select **Auto-check Sediment Intermediate Result**. In **Set Time Interval** dialog, set the time interval as 10 and then click **OK**.

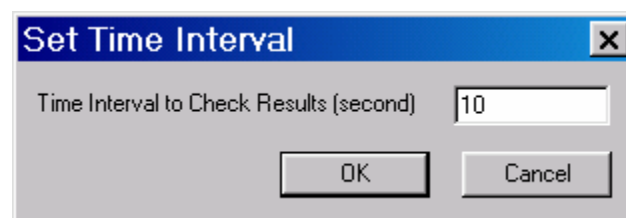


Figure 3-23

- After the simulation is finished, you can visualize the final results file.
 - Select **Sediment Final Results File** in **Visualization** menu.
 - In **Select Sediment Results File** dialog window, select the flow field and then click **OK**.

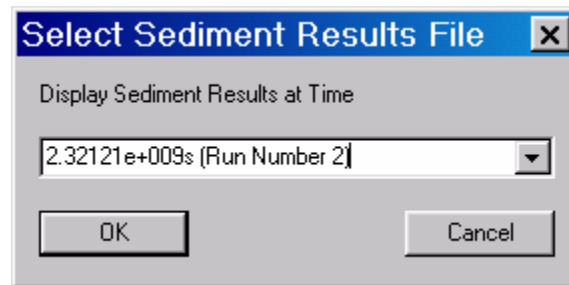


Figure 3-24

- After a result file (**mds** or **sdm**) is loaded,
 - You can select sediment variables from **variable selector** on the main toolbar.

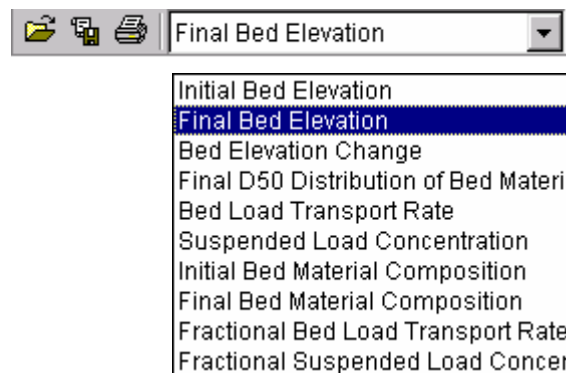


Figure 3-25

- If you select **Fractional Bed Load Transport Rate** or **Fractional Suspended Load Concentration**, you can select **Sediment Size Class Display...** in menu **Visualization** to view the results for each size class.

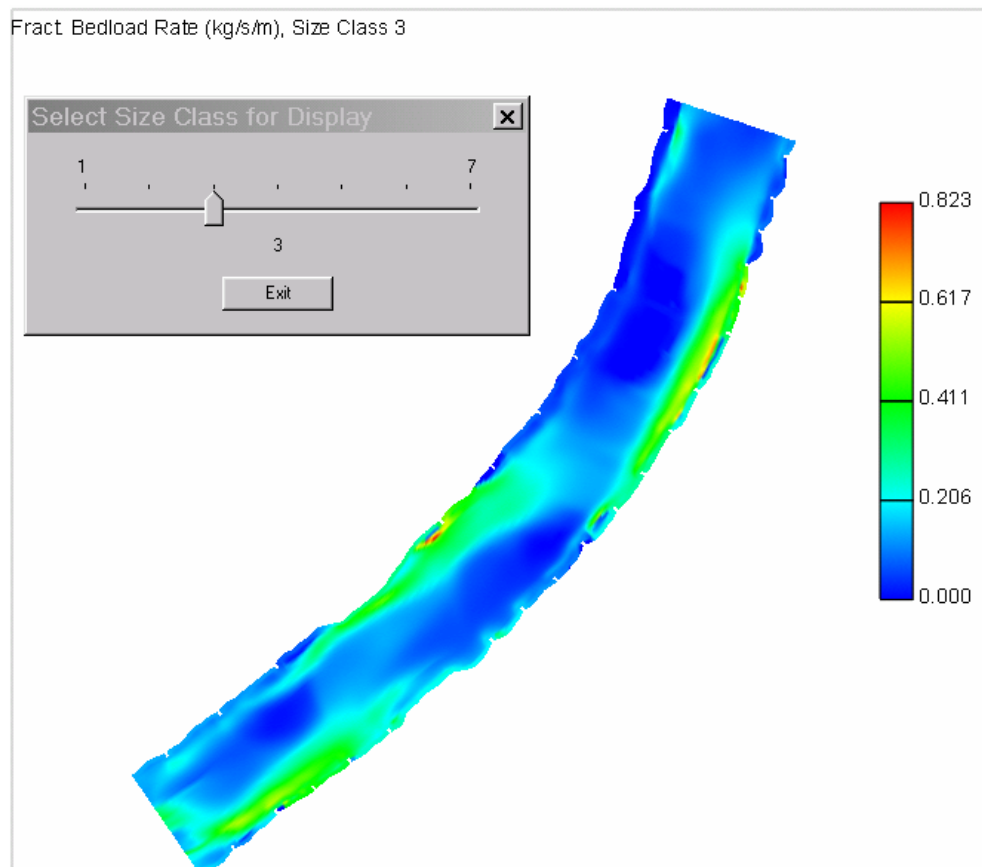


Figure 3-26

- After the simulation is finished, you can use the **History File Editor** to manipulate the visualization of the sediment history results. (Please see section 2.7)

4 Example 2: Unsteady Flow Simulation Using Simulation Wizard


4.1 Introduction

This chapter will illustrate how to use CCHE2D-GUI to simulate unsteady flow without sediment transport using the data from Esfork River, USA. All data is included as example file in the installation package of CCHE2D-GUI.

The simulation procedure for unsteady flow without sediment transport is quite similar to Vistula case (see Chapter 2). To avoid repeating, this chapter will focus on the use of wizard to run simulations.

4.2 Open Geo File

The first step for setting up a case for simulation is to open a geo file.

- Start CCHE2D-GUI by either double-clicking the CCHE2D-GUI icon on your Windows desktop or click the icon from CCHE2D-GUI group of your Windows programs.
- Select **Open Geometry...** in **File** menu or click  in the main toolbar. Navigate to the directory of **Esfork** and select the **Esforkh1.geo**, then click **Open**.

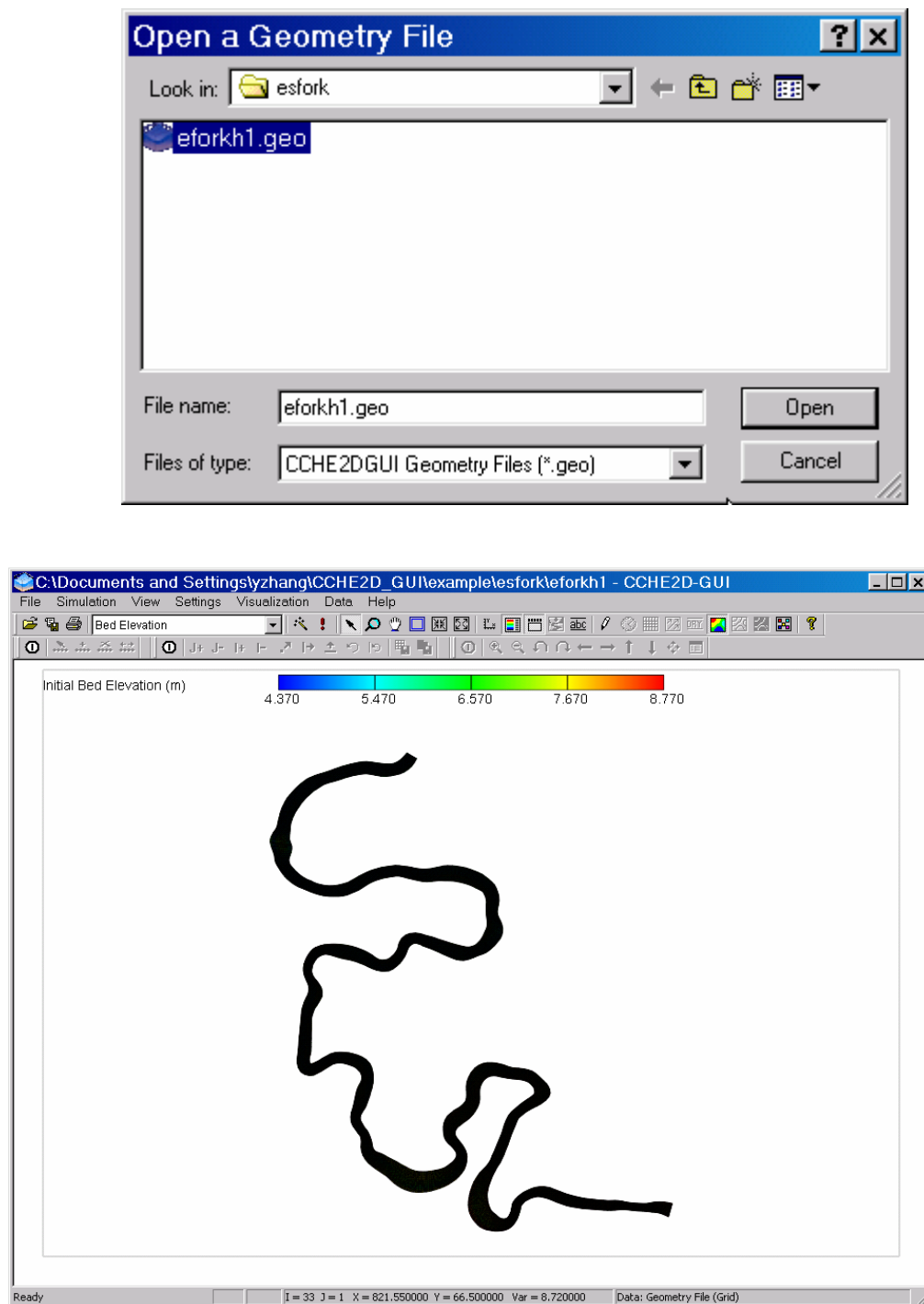



Figure 4-1

4.3 Use Simulation Wizard

To help the first-time users quickly master how to run simulations through CCHE2D-GUI, a simple wizard is provided.

- First select **Wizard** in menu **Simulation** or click  in the main toolbar to activate the **Wizard** window. There are 9 steps listed, among which steps 4, 5, 7, and 8 are related to the sediment transport simulation. In this case, we will skip the above steps.

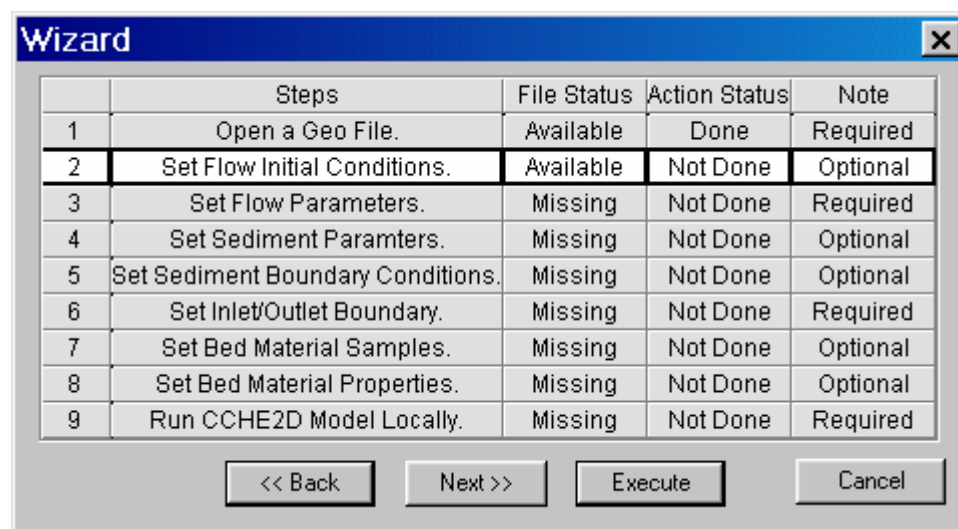


Figure 4-2

- Since the first step **Open a Geo File** is already done, we will begin with step 2 **Set Flow Initial Conditions**.
 - Use **Next >>** button to navigate to step 2 **Set Flow Initial Conditions**, and then click **Execute**. The Nodal Properties dialog window will appear.
 - In **Nodal Properties** dialog, select **Initial water surface level** and then click **Whole Domain**. In **Assign Value** dialog, enter the value 8.0, and then click **OK**.

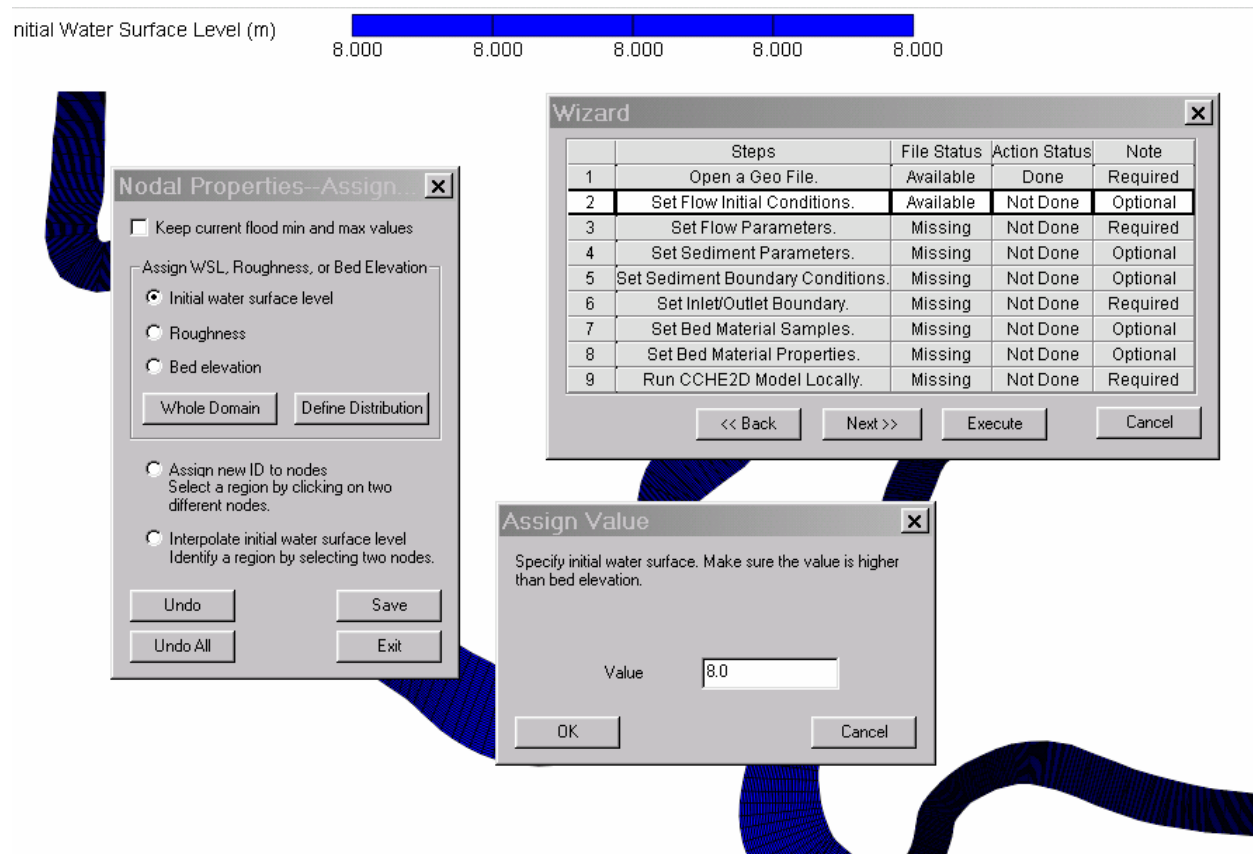


Figure 4-3

- In **Nodal Properties**, select **Roughness** and then click **Whole Domain**. In **Assign Value**, enter the value of 0.03, and then click **OK**.

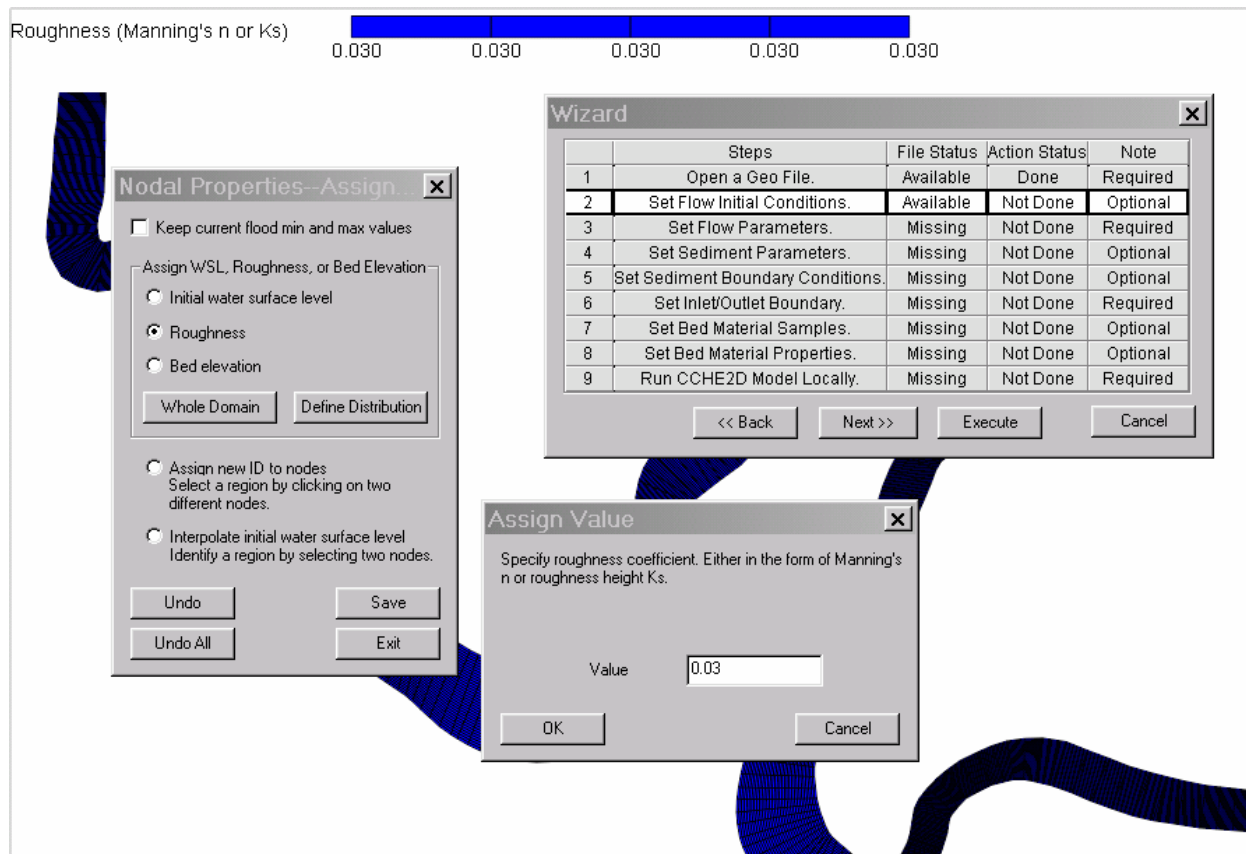


Figure 4-4

- In **Nodal Properties**, click **Save** to save your changes. Then click **Exit**. The **Action Status** of step 2 will become “**Done**”.
- Then we go to step 3 **Set Flow Parameters** using **Next >>** button.
 - Click **Execute** to activate the **Set Flow Parameters** dialog window.
 - In Simulation Parameters page, we set the simulation time as 2592000 s (= 30 days), and set the time step as 600 s (=10 mins). Other parameters are shown in Figure 3-4.

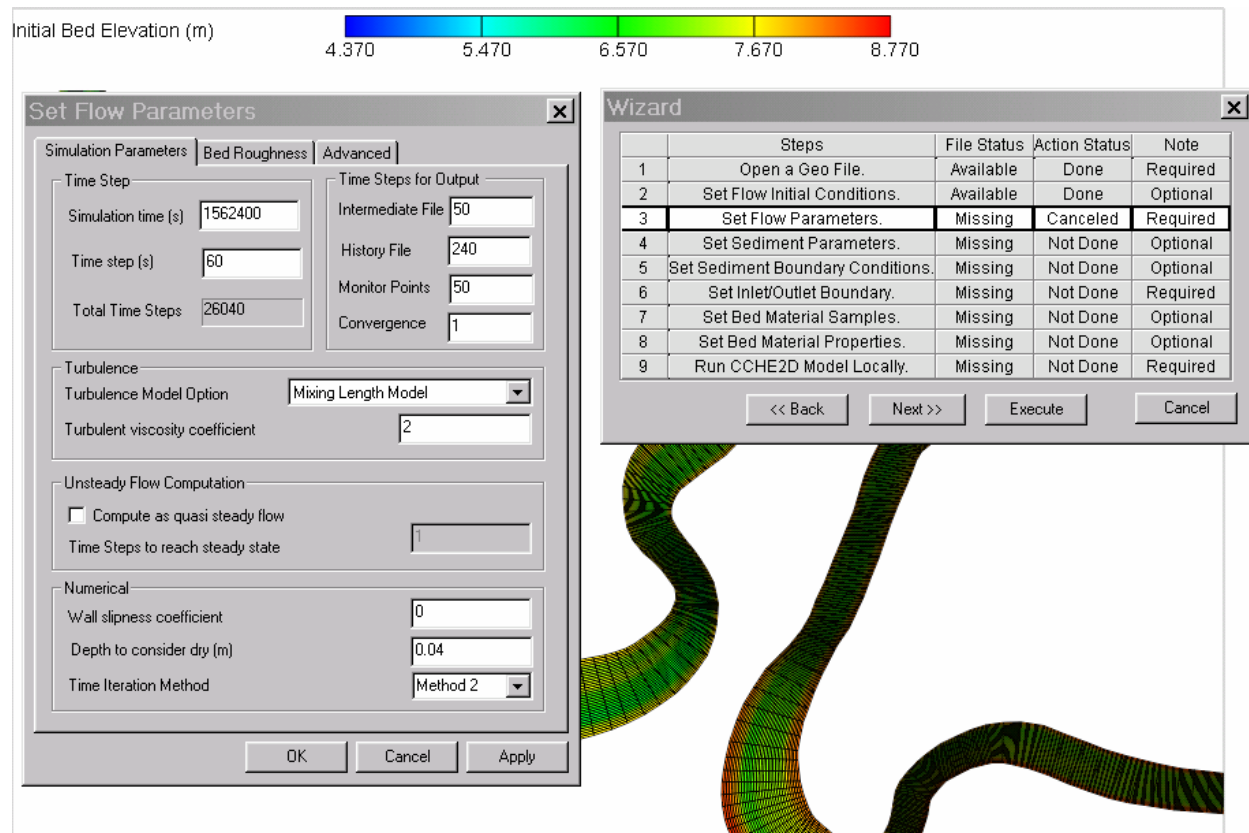


Figure 4-5

- For parameters in **Bed Roughness** page and **Advanced** page, we use the default values.
- In **Set Flow Parameters**, click **OK** to save the parameters.
- We skip steps 4 and 5 and go to step 6 **Set Inlet/Outlet Boundary Conditions**.
 - Click **Execute** to activate the boundary editing toolbar
 - Click two ending points along the first mesh J line and define it as inlet boundary. Select **Inlet Boundary Condition** in **Select Inlet/Outlet Boundary** dialog window, and then click **OK**. The **Inlet Boundary Conditions** dialog window will appear.

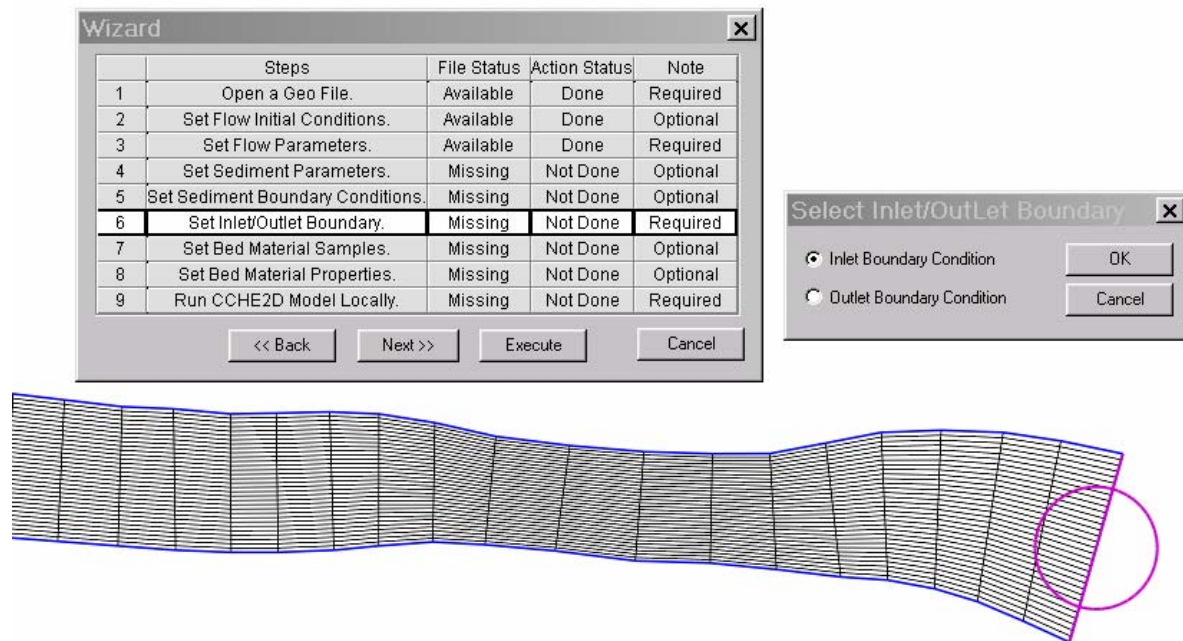


Figure 4-6

- In **Inlet Boundary Conditions**, in **Flow** page, select option **Discharge hydrograph**. In the **Open a Discharge Hydrograph File** dialog, choose **Esforkh1.dhg** and then click **Open**. Then click **OK** in **Inlet Boundary Conditions** to save the boundary conditions. Note that the discharge hydrograph must be provided by the user, and its format can be found in CCHE2D-GUI User's Manual. For this case, the file **Esforkh1.dhg** is include in the Esfork River example. **Note that the hydrograph must start from $t = 0$.**

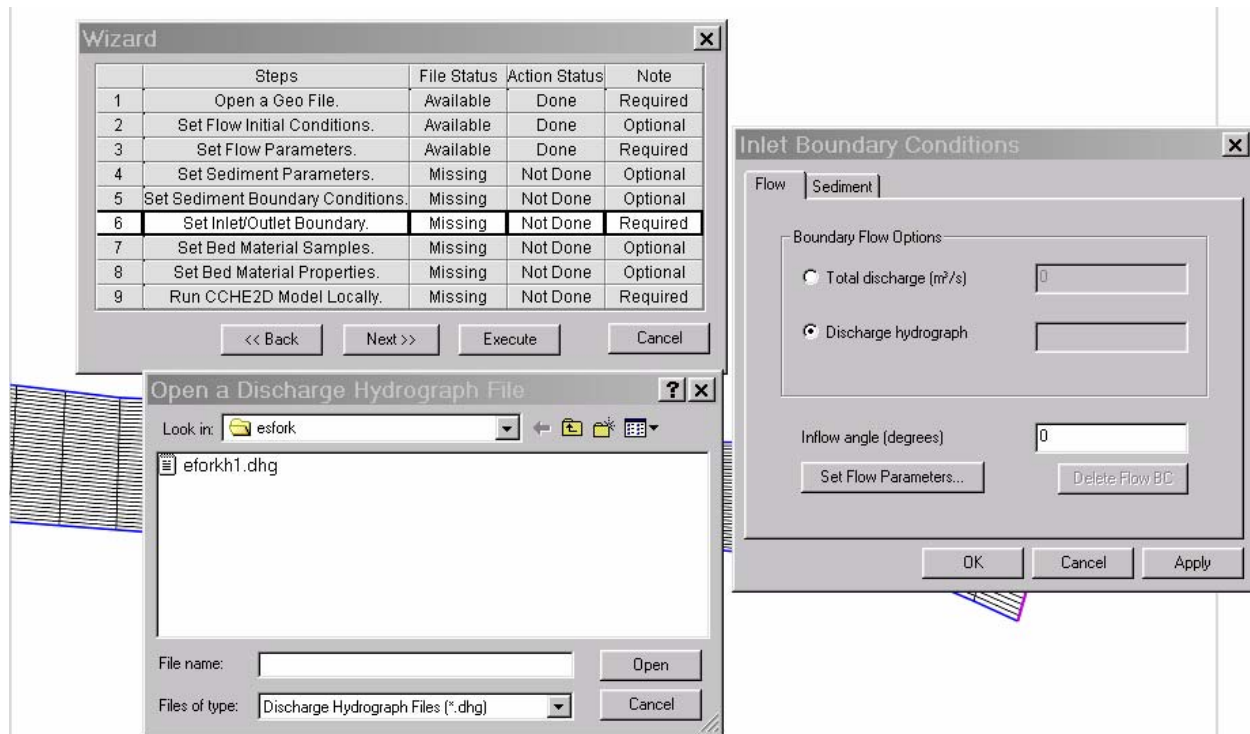


Figure 4-7

- Click two ending points along the last mesh J line and define it as outlet boundary. Select **Outlet Boundary Condition** in **Select Inlet/Outlet Boundary** dialog window, and then click **OK**. The **Outlet Flow Boundary Conditions** dialog window will appear.

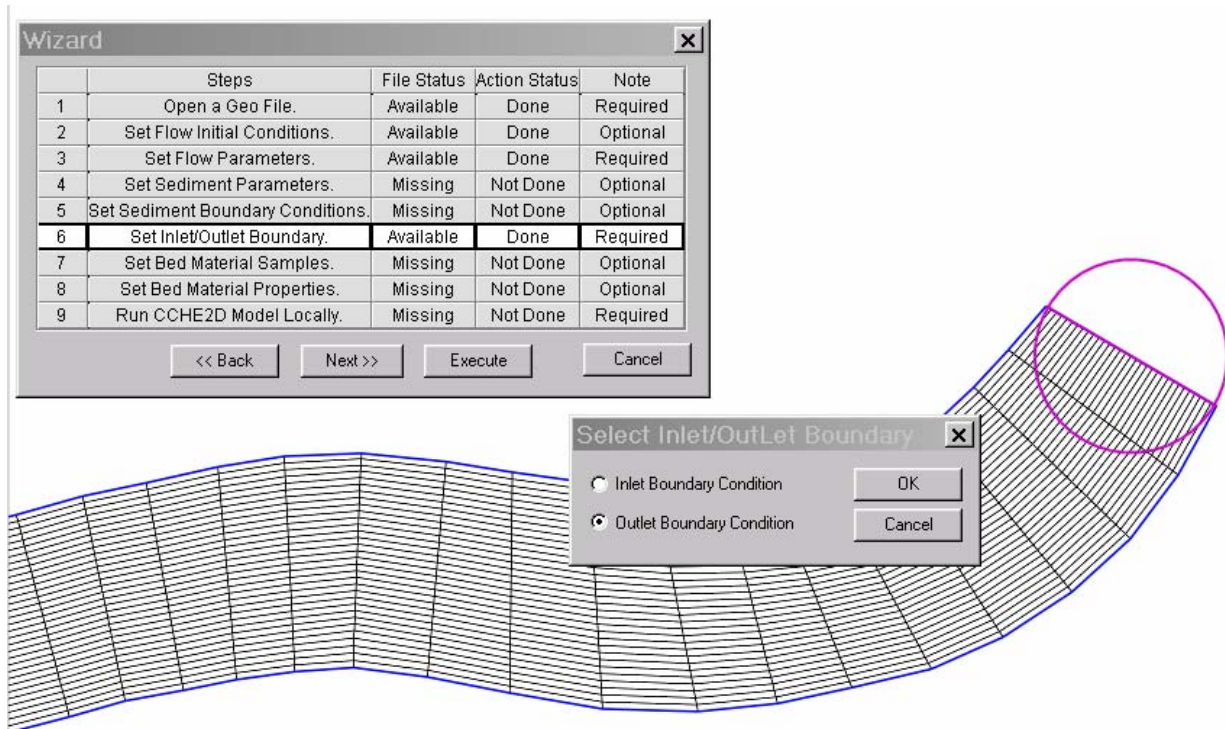


Figure 4-8

- In **Outlet Flow Boundary Conditions**, select option **Stage hydrograph**. In the **Open a Stage Hydrograph File** dialog, choose **Esforkh1.shg** and then click **Open**. Then click **OK** in **Outlet Flow Boundary Conditions** to save the boundary conditions. Similarly, the stage hydrograph file must be provided by the user, and its format can be found in CCHE2D-GUI User's Manual. For this case, the file **Esforkh1.shg** is include in the Esfork River example.

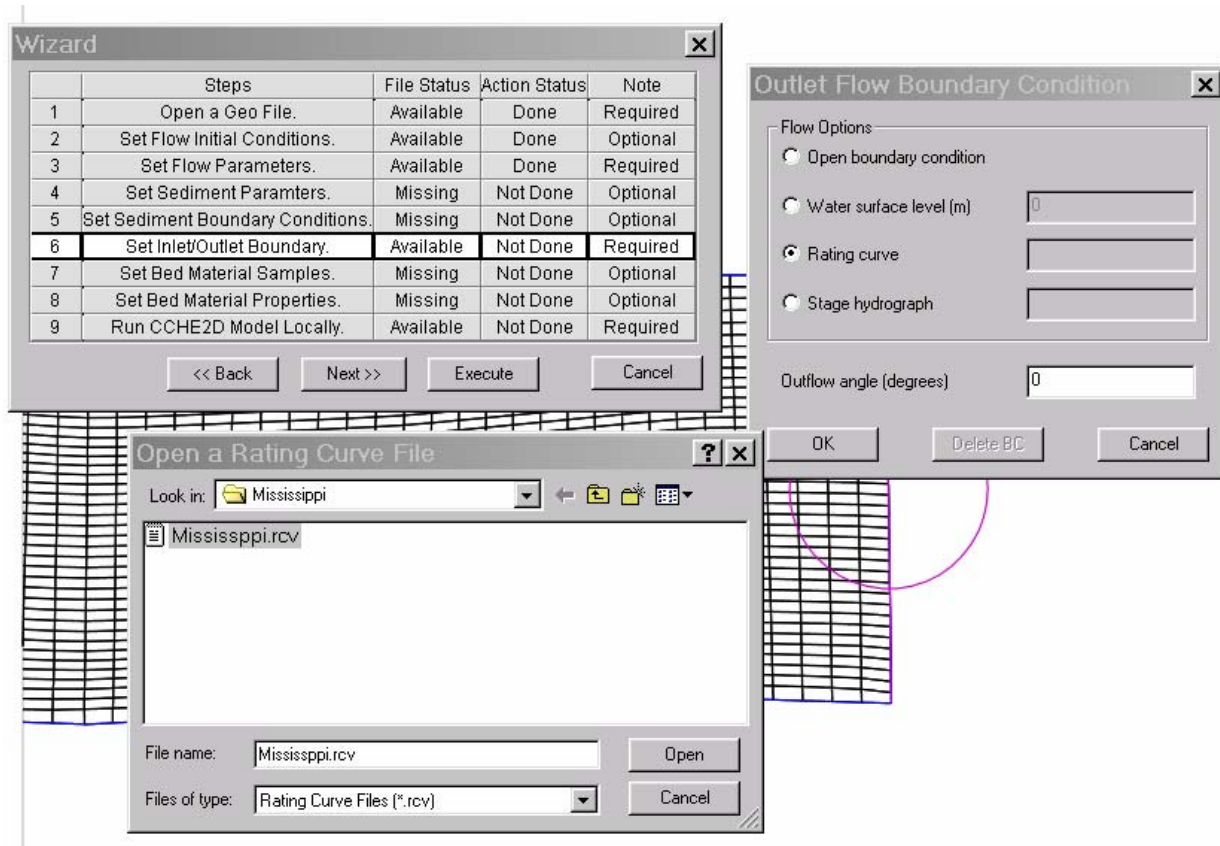
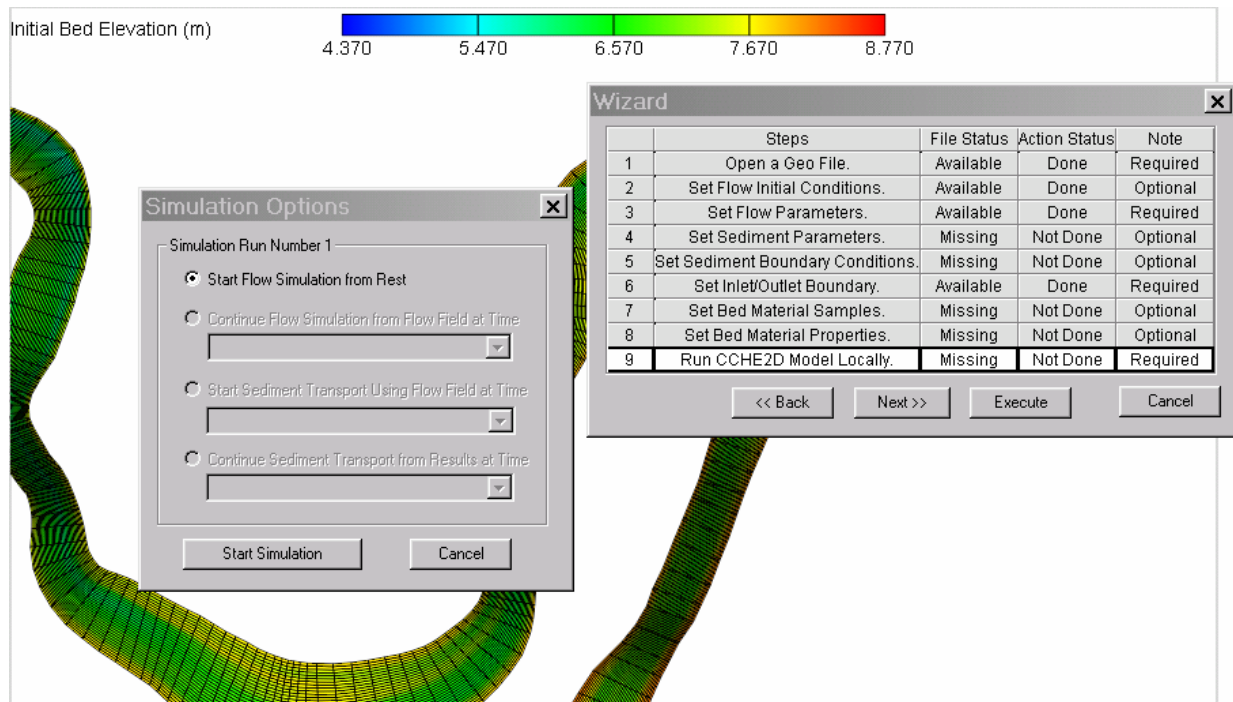


Figure 4-10

- We skip steps 7 and 8, and go to step 9 **Run CCHE2D Model Locally**.
 - Click **Execute** to activate the **Simulation Options** dialog window.
 - In **Simulation Options** dialog, select option **Start Flow Simulation from Rest**, and then click **Start Simulation**.



```

C:\WINNT\system32\cmd.exe
159 3.680556 % of the total timesteps
160 3.703704 % of the total timesteps
161 3.726852 % of the total timesteps
162 3.750000 % of the total timesteps
163 3.773148 % of the total timesteps
164 3.796296 % of the total timesteps
165 3.819444 % of the total timesteps
166 3.842592 % of the total timesteps
167 3.865741 % of the total timesteps
168 3.888889 % of the total timesteps
169 3.912037 % of the total timesteps
170 3.935185 % of the total timesteps
171 3.958333 % of the total timesteps
172 3.981482 % of the total timesteps
173 4.004630 % of the total timesteps
174 4.027778 % of the total timesteps
175 4.050926 % of the total timesteps
176 4.074074 % of the total timesteps
177 4.097222 % of the total timesteps
178 4.120370 % of the total timesteps
179 4.143518 % of the total timesteps
180 4.166667 % of the total timesteps
181 4.189815 % of the total timesteps
182 4.212963 % of the total timesteps
183 4.236111 % of the total timesteps
  
```

Figure 4-11

4.4 Visualize Results

As stated in chapter 2, there are three kinds of flow results files: flow intermediate file (*.mdw), flow final results file (*.flw) and flow history file (*.his).

- During the simulation, you can view the intermediate results file. You can choose to load it either manually or automatically. (Please see section 2.11)

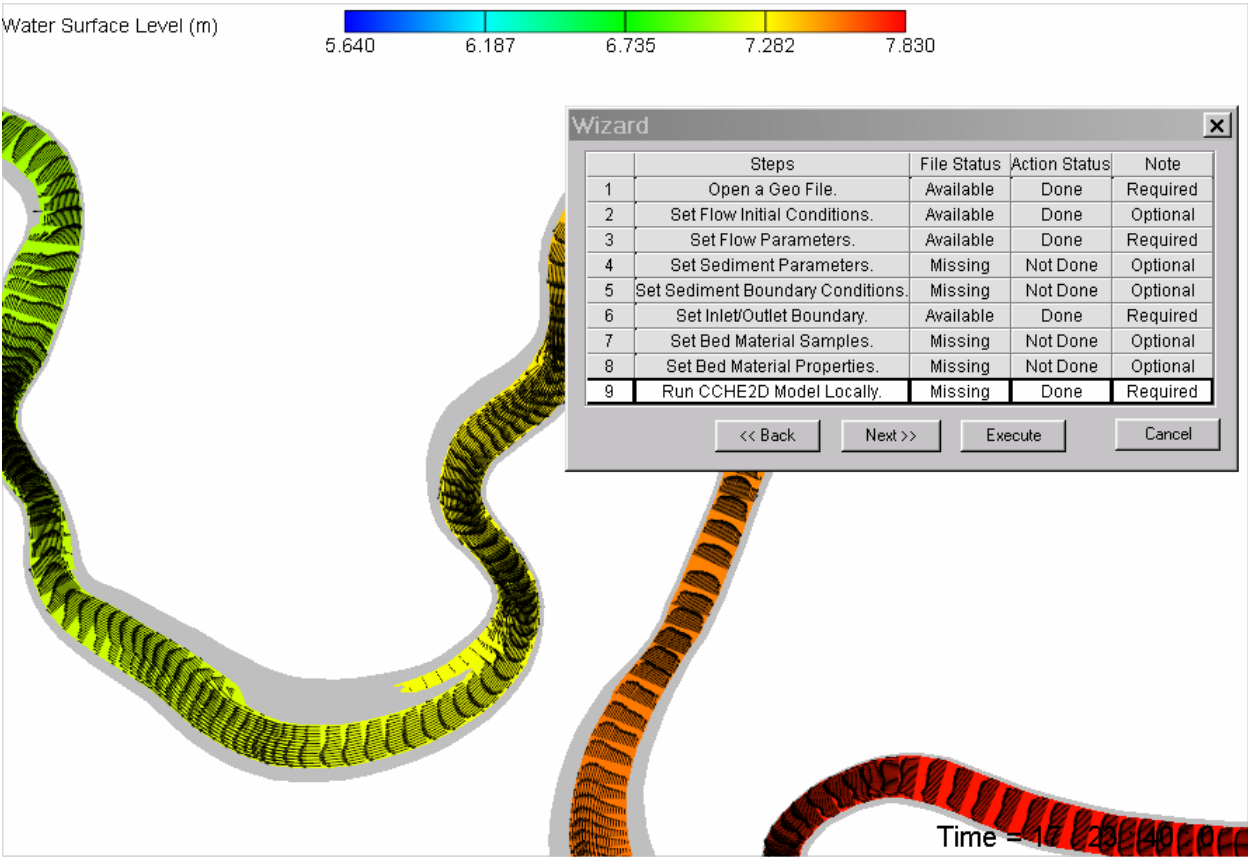


Figure 4-12

- After the simulation finished, you can view the flow final results file and the history file. (Please see section 2.11)

References

- Jia, Yafei and Wang, Sam S.Y. (2001). “CCHE2D: Two-dimensional Hydrodynamic and Sediment Transport Model for Unsteady Open Channel Flows Over Loose Bed.” NCCHE Technical Report, NCCHE-TR-2001-01, Aug 2001.
- Wu, Weiming (2001). “CCHE2D Sediment Transport Model.” NCCHE Technical Report, NCCHE-TR-2001-03, Aug 2001.
- Zhang, Yaixin (2005). “CCHE2D Mesh Generator Users’ Manual—Version 2.50.” NCCHE Technical Report. NCCHE-TR-2005-01, Jan 2005.
- Zhang, Yaixin (2005). “CCHE2D-GUI: Graphical User Interface for CCHE2D Model. Users Manual—Version 2.2.” NCCHE Technical Report. NCCHE-TR-2005-03, May 2005.